DOCUMENT RESUME

ED 316 756 CE 054 454

AUTHOR Crosswhite, Dwight
TITLE Machine Shop. Module 8: CNC (Computerized Numerical Control). Instructor's Guide.
INSTITUTION Missouri Univ., Columbia. Instructional Materials Lab.
SPONS AGENCY Missouri State Dept. of Elementary and Secondary Education, Jefferson City. Div. of Vocational and Adult Education.
PUB DATE 89
NOTE 138p.; For related documents, see CE 054 447-453. Document contains colored paper. Student reference and worksheets are included in this document.
AVAILABLE FROM Instructional Materials Laboratory, University of Missouri, 2316 Industrial Drive, Columbia, MO 65202 (order no. 70-6178-I: $14.25).
PUB TYPE Guides - Classroom Use - Guides (For Teachers) (052)

EDRS PRICE MF01 Plus Postage. PC Not Available from EDRS.
DESCRIPTORS *Automation; Competency Based Education; Educational Objectives; Electronic Control; *Machinery Industry; *Machine Tool Operators; *Machine Tools; Machinists; Measurement Equipment; *Numerical Control; Occupational Information; *Occupational Safety and Health; Postsecondary Education; Secondary Education; Tool and Die Makers; Trade and Industrial Education
IDENTIFIERS Missouri

ABSTRACT

This document consists of materials for a five-unit course on the following topics: (1) safety guidelines; (2) coordinates and dimensions; (3) numerical control math; (4) programming for numerical control machines; and (5) setting and operating the numerical control machine. The instructor's guide begins with a list of competencies covered in the module, descriptions of the materials included, an explanation of how to use the materials, and information on how the curriculum guide is packaged. Next is a table that associates individual instructional materials in the units with the competencies they teach. The instructional units follow, each of which may include unit objectives, lesson objectives, lesson plans, references, notes to the teacher, information sheets or outlines, transparency masters, handouts, assignment sheets, job sheets, unit tests, and answers to tests. (CML)

****
* Reproductions supplied by EDRS are the best that can be made *
* from the original document. *
****

ED316756

70-6178-I

MACHINE SHOP CURRICULUM

U.S. DEPARTMENT OF EDUCATION
Office of Educational Research and Improvement
EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)

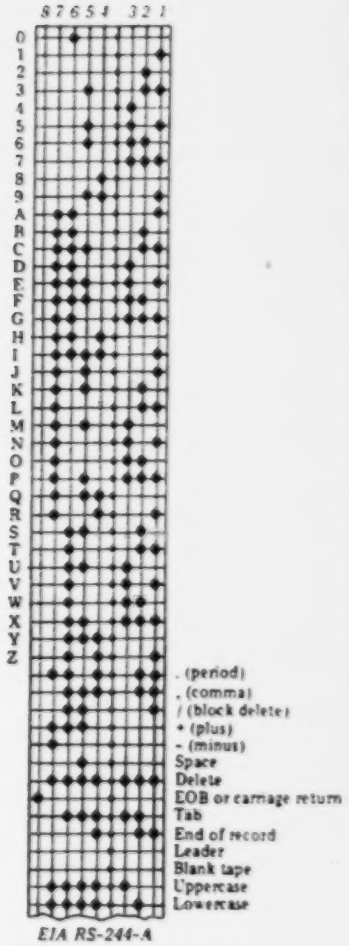
This document has been reproduced as received from the person or organization originating it.

Minor changes have been made to improve reproduction quality.

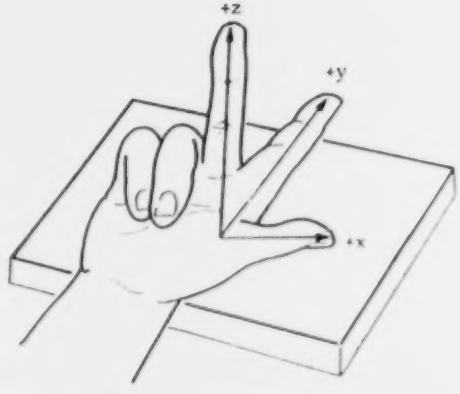
Points of view or opinions stated in this document do not necessarily represent official OERI position or policy.

INSTRUCTOR'S GUIDE

MODULE 8: CNC (Computerized Numerical Control)


8 7 6 5 4 3 2 1
0 1 2 3 4 5 6 7 8 9 A B C D E F G H I J K L M N O P Q R S T U V W X Y Z
. (period)
, (comma)
/ (block delete)
+ (plus)
- (minus)
Space
Delete
EOB or carriage return
Tab
End of record
Leader
Blank tape
Uppercase
Lowercase

EIA RS-244-A format



"PERMISSION TO REPRODUCE THIS MATERIAL IN MICROFICHE ONLY HAS BEEN GRANTED BY

J. Schlichting

TO THE EDUCATIONAL RESOURCES INFORMATION CENTER (ERIC)."

INSTRUCTIONAL MATERIALS LABORATORY • UNIVERSITY OF MISSOURI-COLUMBIA • COLUMBIA, MISSOURI

CE 054 454

Modules in this IML Machine Shop series

1: *Machine Shop Orientation and Math* by Donna Curtis and Jack Nobles
2: *Measurement and Blueprint Reading* by Charles H. Walden, Don Marquardt and Henry "Junior" Cagle
3: *Bench Work and Material Science* by Charles H. Walden and Jack Nobles
4: *Power Saw and Drill Press Operation* by Charles H. Walden and Bill Daniel
5: *Lathes* by Jack Nobles
6: *Milling* by Charles H. Walden
7: *Grinders* by Jack Nobles and Mel Gage
8: *CNC (Computerized Numerical Control)* by Dwight Crosswhite

The activity which is the subject of this report was supported in whole or in part by funds from the Department of Elementary and Secondary Education, Division of Vocational and Adult Education. However, the opinions expressed herein do not necessarily reflect the position or policies of the Missouri Department of Elementary and Secondary Education or the Division of Vocational and Adult Education, and no official endorsement should be inferred.

MACHINE SHOP

Module 8: CNC (Computerized Numerical Control)

by

Dwight Crosswhite

Diane M. Davis, editor/project coordinator
Instructional Materials Laboratory
8 London Hall
University of Missouri-Columbia
Columbia, MO 65211

1989

Instructor's Guide
Catalog No. 70-6178-I

FOREWORD

Once again, we are indebted to the teachers and administrators who provide their time, efforts and professionalism to develop curriculum for trade, technical and industrial education programs. This Machine Shop guide is an outstanding example of what can be accomplished when the right people with the right attitude get together. Committee members worked hard over several years' time in advisory, writing, technical consultation and field-test capacities.

This guide is designed to be totally integrated with the Vocational Instructional Management System. It is hoped that all guides produced under our supervision will help classroom teachers manage instruction. A curriculum guide is more than a resource--it is an integral part of the management system. Rather than impeding instruction, it should guide and facilitate instruction.

This guide's format has been set up for competency-based teaching and testing. Unit tests evaluate specific competencies and can be used with the Vocational Administrative Management System (VAMS). Information sheets, transparency masters, job sheets, assignment sheets and test items are keyed to specific competencies. A cross-reference table has been included to quickly identify parts of the curriculum that relate to specific competencies. The material has also been packaged for students.

Although this guide may be a new format for some teachers, we are confident that, when used as designed, time spent by teachers and students will become more productive. Any future machine shop curriculum modifications will begin with this guide. The machine shop field is undergoing considerable change. We will annually evaluate the need to update this guide on a module-by-module basis. Your suggestions regarding areas for improvement are both encouraged and appreciated.

Judith Moore, supervisor
Industrial Education
Department of Elementary and Secondary Education

Charles "Chuck" Waibel, director
Industrial Education
Department of Elementary and Secondary Education

ACKNOWLEDGMENTS

This Machine Shop guide represents a continuing commitment to Missouri's Vocational Instructional Management System (VIMS). The guide has been keyed to the machine shop competency profile developed by industry and education professionals in the state.

Appreciation is given to the writers (Charlie Walden, Jack Nobles, Dwight Crosswhite, Donna Curtis, Don Marquardt, Bill Daniel, Mel Gage and Jr. Cagle) for their technical expertise. Catherine Smith of Dodge Art Works, Columbia, created a large portion of the guide's artwork.

These industry representatives and educators contributed to the guide as members of the advisory committee.

Junior Cagle, Poplar Bluff AVTS
Bill Daniel, Hannibal AVTS
Mel Gage, Linn Technical College
Pete Lopez, N.S. Hillyard AVTS, St. Joseph
Jack Nobles, McDonnell-Douglas Aircraft, St. Louis
Charlie Walden, Mechanical & Aerospace Engr., University of Missouri-Columbia
Mike Dyrenfurth, Dept. of PAVTE, University of Missouri-Columbia
Charles Waibel and Judith Moore, Division of Vocational & Adult Education, Department of Elementary and Secondary Education, Jefferson City

Educators who field-tested the guide in the classroom include the following.

Junior Cagle, Poplar Bluff AVTS
Bill Daniel, Hannibal AVTS
Charles Dawson, Moberly AVTS
Mel Gage, Linn Technical College
Pete Lopez, N.S. Hillyard AVTS, St. Joseph
David Michael, Lex La-Ray Area Vocational Technical Center, Lexington
James Newman, Jefferson College, Hillsboro
Willburn "Kirt" Salmons, State Fair Community College, Sedalia
Russell Schleiermacher, Davis H. Hart Mexico AVTS
James Shimel, Longview Community College, Lee's Summit
Coy Smith, Rolla AVTS

Support and contributions of IML staff members were instrumental to this module's development.

Harley Schlichting, director
Amon Herd, associate director
Phyllis Miller and Dan Stapleton, assistant directors
Sharon Dyer, graphic artist
Donna Curtis and Lori Holliday, word processors
Julie Bishop, student assistant - clerical
Wei Li, graduate assistant

TABLE OF CONTENTS

Unit I: Safety guidelines 1

Unit II: Coordinates and dimensions 33

Unit III: NC math 75

Unit IV: Programming for NC machines 103

Unit V: Setting and operating the NC machine 147

COMPETENCIES ADDRESSED IN THIS MODULE

M. Computerized Numerical Control (CNC) Machines

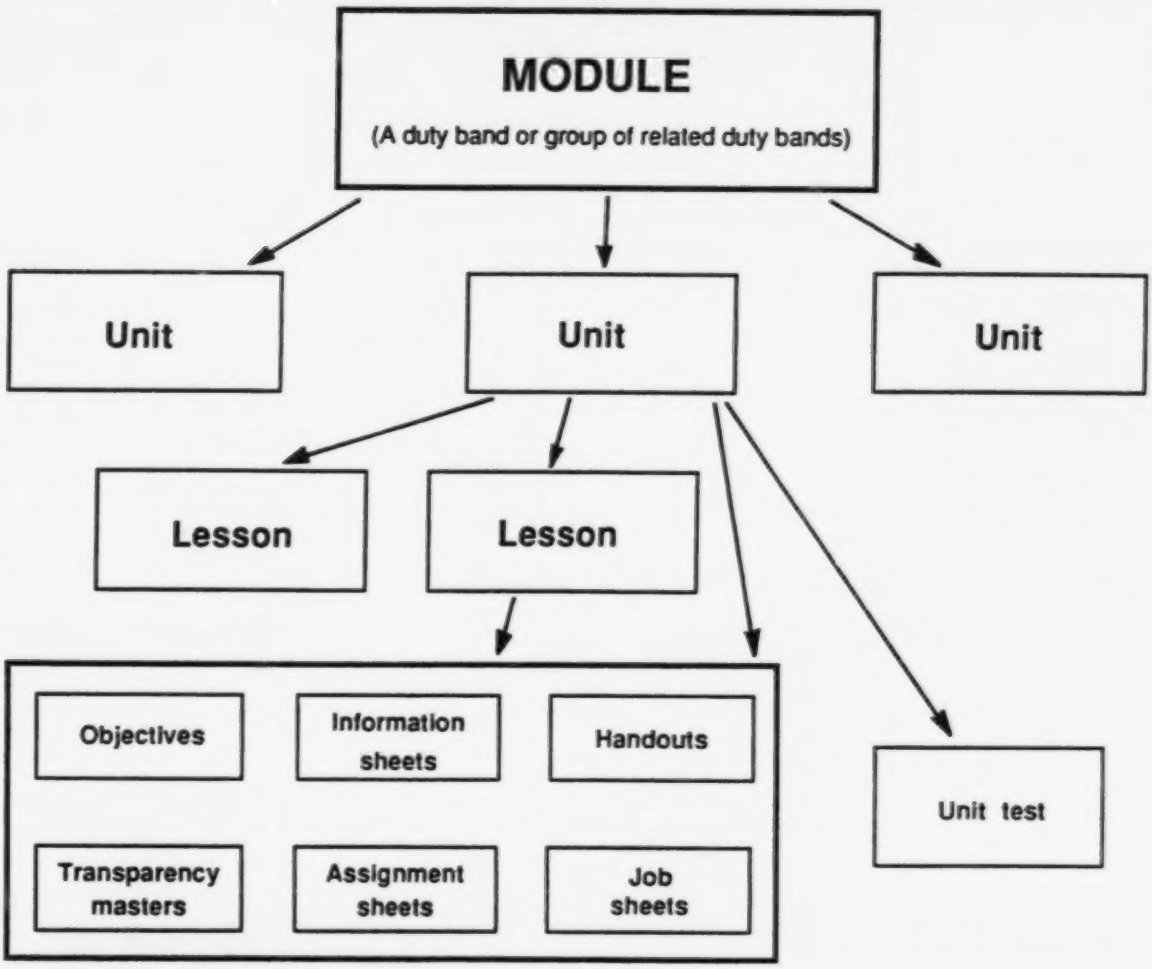
1. Employ CNC machine safety guidelines consistently
2. Perform care and maintenance
3. Calculate coordinates and dimensions of CNC drawing
4. Write program for CNC machine
5. Set up a CNC machine
6. Machine workpiece with CNC machine

HOW TO USE THIS PUBLICATION

GUIDE COMPONENTS

Units of instruction

This Machine Shop guide contains eight modules of instruction. Within each module are instructional units including some or all of the following components: performance objectives, information sheets, transparency masters, handouts, assignment sheets, job sheets and a unit test. Special instructions on audiovisuals, supplies, etc. are included in the Notes to the Instructor section near the beginning of each unit.



Competency profile

On the Missouri competency profile, tasks are divided into general subject areas, or duty bands, which are identified by a capital letter. Under each duty band are related tasks (competencies), which are identified by an Arabic numeral. To evaluate the student on performance of the competencies, a 3-2-1-N grading system is used, where:

- **3 = Mastery** - can work independently with no supervision
- **2 = Requires supervision** - can perform job completely with limited supervision
- **1 = Not mastered** - requires instruction and close supervision
- **N = No exposure** - no experience or knowledge in this area

Cross-reference table (color-coded white)

Once the numbering system is understood, this table quickly reveals how the instructional components of the module relate to Missouri competencies. On the competency profile, tasks are divided into general subject areas, or duty bands, which are identified by a capital letter. Under each duty band are related tasks (competencies), which are identified by an Arabic numeral.

On the cross-reference table, unit numbers are represented by Roman numerals (I, II, III, etc.). Lessons within a unit are shown with Arabic numerals. Therefore, the first objective in Lesson 1 is numbered 1.1, the second is 1.2, and so on. Other components of the module (transparencies, handouts, assignment sheets and job sheets) also use this numbering system. By looking at the cross-reference table, the instructor can quickly determine what test questions or job sheets a student needs to receive an evaluation on a particular competency.

Objectives (color-coded white)

Each unit is based on performance objectives which state the measurable unit and specific behavioral or performance objectives that students are expected to achieve. Since the objectives of the unit provide direction for the teaching-learning process, it is important for the teacher and students to have a common understanding of the objectives' intent. Teachers should feel free to add/adapt objectives which will fit the material to student and community needs. Supporting material will, of course, need to be added for any new objectives.

Table of contents, references, notes to the instructor (color-coded pink)

These pages have been coded pink for instructor use only. The reference section lists materials used as sources in developing the material. Special instructions on audiovisuals, supplies, etc. are included in the Notes to the Instructor section.

Information sheets (green)

Presented in outline form for clarity, these pages provide content essential for meeting the cognitive (knowledge) objectives in the unit. Students should study the information sheets before class discussion or completion of assignment sheets. **CAUTION** statements indicate possible equipment damage or bodily injury. Handouts and transparency masters have been keyed in the information sheets for instructional use.

Transparency masters (white)

Transparencies provide information by visually reinforcing or clarifying ideas presented in the information sheets. They are effective when identification is necessary and direct the class's attention to the topic of discussion. Transparencies should be left on the screen only when topics shown are under discussion, however.

Handouts (white)

Whenever more detailed information than can be presented in a transparency is needed, a handout is used. Handouts give the students additional information that can be kept with their class notes or student references.

Assignment sheets (tan)

Assignment sheets furnish paper-and-pencil practice to develop the knowledge necessary as a prerequisite to skill development. They can be given to students for class completion or used as homework assignments. Answer sheets (color-coded pink) can be used by the student or instructor to check progress.

Job sheets (blue)

These evaluation instruments are provided for two purposes: to outline steps of procedure and to check student performance mastery. Job sheets begin with a list of tools and materials needed for the activity. Job sheets furnish potential employers with a picture of skills being taught and performance which might be reasonably expected from a vocational student. The spaces to the right of the page on a job sheet

assist the instructor in evaluating a student on a particular activity. This 3-2-1-N grading system is the same as the one used on the competency profile.

Unit test (yellow)

Competency-based tests are designed to help eliminate subjective testing and to improve student evaluation and accountability. Specific competencies and test questions are linked on the answer key. These unit tests have been written to be used with computerized VAMS (Vocational Administrative Management System), utilizing Scantron sheets for computer-recording of test answers.

The unit test may be eliminated if the competency was thoroughly evaluated by job sheets. However, an emphasis should be placed on cognitive as well as performance skills. If the objective or competency will be evaluated by a job sheet, it is often necessary to determine by a written test whether the student understands enough of the material to perform the job sheet.

SUGGESTED ACTIVITIES

The following list of suggested activities may be helpful, especially to the beginning instructor.

1. Provide students with objective sheet and information sheets.
2. Discuss unit and specific objectives.
3. Discuss information sheets, assignment sheets and/or job sheets, lesson-by-lesson.
4. Have students do assignment sheet(s) to satisfactory levels.
5. Give students unit test.
6. After successful completion of the test, have students do job sheet(s).

Students should perform the following activities.

1. Read objective sheet.
2. Study information sheets.
3. Do assignment sheet(s).
4. Take unit test.
5. Do job sheets.

PACKAGING

This Machine Shop guide has been packaged with the entire class in mind. For the teacher, a complete instructor's guide contains all eight modules. In the student reference, selected portions have been reproduced for student use (objectives page, information sheets, handouts). Student work sheets (assignment sheets, job sheets, unit tests) have been packaged together for student distribution as the instructor deems necessary.

In the student reference, page numbers do not run consecutively because some pages have been removed that appear only in the instructor's guide.

MODULE 8: CNC (COMPUTERIZED NUMERICAL CONTROL)

CROSS-REFERENCE TABLE OF INSTRUCTIONAL MATERIALS

Competency	Instructional Objective	Transparency Master	Handout	Assignment Sheet	Job Sheet	Test Item	Notes
M1	I 3.1-3.2			I 3.1		I 8,15-16,19-26	
M2	V 2.1					V 20	V instructor note
M3	II 1.1-2.1		II 1.1-1.4	II 1.1,1.2,2.1		II 8,9,13-15,28-35	
M4	II 3.1-3.3, III 1.1,1.2, IV 1.1-4.1	III 1.1,1.2	III 1.1-1.3, IV 2.1-2.4	II 3.1, III 1.1, IV 1.1,1.2, 2.1,3.1		II 19-23, III 1,3-6,9-21, IV 1-26	
M5	V 1.1-1.3		V 1.1-1.3	V 1.1		V 1-6,8,9,11-17	
M6	V 2.2-3.1		V 3.1			V 7,10,18,19	

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)
UNIT I: SAFETY GUIDELINES

UNIT OBJECTIVE

Upon completion of this unit, the student should be able to identify the safety guidelines and principles of numerically controlled machining. This knowledge will be evidenced by achieving the specified score on the assignment sheets and unit test.

SPECIFIC OBJECTIVES

After completion of the lessons in this unit, the student should be able to:

1.1 Identify true statements concerning NC (numerical control) history.

1.2 Solve problems concerning advantages and disadvantages of NC.

1.3 Match basic NC terms with definitions.

2.1 List the basic components of the NC system.

2.2 Define specific NC terms.

2.3 Identify NC's part in the overall manufacturing plan.

3.1 Match terms concerning general safety rules.

3.2 Identify specific NC safety rules. (competency M1)

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)

UNIT I: SAFETY GUIDELINES

CONTENTS OF THIS UNIT

A. Objective sheet

B. References

C. Notes to the instructor

D. Lesson plans

1. Lesson 1: ORIENTATION

   a. Informational outline

   b. Assignment sheet

      AS 1.1: CNC orientation

   c. Answers to assignment sheet

2. Lesson 2: HOW NUMERICAL CONTROL WORKS

   a. Informational outline

   b. Handouts

      HO 2.1: How NC fits into the manufacturing plan
      HO 2.2: The scope of responsibility (simplified)

   c. Assignment sheet

      AS 2.1: NC procedures

   d. Answers to assignment sheet

3. Lesson 3: SAFETY

   a. Informational outline

   b. Assignment sheet

      AS 3.1: NC safety

   c. Answers to assignment sheet

E. Unit test

F. Answers to unit test

REFERENCES

A. Books

1. Donaldson, Cyril, George H. LeCain and V. C. Goold. Tool Design. 3rd ed. New York: McGraw-Hill, 1973.

2. Feirer, John L. Machine Tool Metalworking Principles and Practice. 2nd ed. New York: McGraw-Hill, 1973.

3. Kibbe, Richard R., et. al. Machine Tool Practices. New York: John Wiley and Sons, 1979.

4. Roberts, Arthur D. Programming for Numerical Control Machines. 2nd ed. New York: McGraw Hill, 1978.

5. Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar Publishers, Inc., 1986.

B. Other

1. Computerized Numerical Control (test item bank). Atlanta, GA: Vocational-Technical Education Consortium of States; Carbondale, IL: Dept. of Vocational Education Studies, Southern Illinois University, 1986.

NOTES TO THE INSTRUCTOR

A. Suggested approaches to teaching this unit

1. The historical value of NC is significant in that machining practices have been revolutionized in a single generation.

2. Note how widespread NC is today.

3. Emphases should be placed on independent thinking and sound logic.

4. Give demonstration of NC machining of a workpiece.

5. Discuss with students the difficulty of machining the part conventionally.

B. Suggested student activity

1. Look for safety violations in the machine shop and show instructor.

C. Suggested supplemental material

1. Introduction to Computer-Aided Manufacturing (MS-84-400) (video). Sedalia, MO: State Fair Community College, 1984. (Available for purchase from SFCC; available for loan from Missouri Vocational Resource Center, phone 800/392-7217 or 314/882-9610)

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)

UNIT I: SAFETY GUIDELINES

LESSON 1: ORIENTATION

I. Brief history of numerical control

A. In 1947 John C. Parsons of Traverse City, Michigan, retrofit a jig bore machine with computer equipment.

B. Two years later, the U.S. Air Force awarded a contract to the Parsons Corporation to expand upon Parsons' idea of integrating a computer-card controlled machine.

C. Soon afterward the Parsons Corporation built the first computer-controlled metal-cutting machine. It was crude by today's standards; nevertheless, it manufactured precision parts.

D. The Air Force entered into an agreement with the Massachusetts Institute of Technology in 1951 to develop the idea further.

E. It was at M.I.T. that the phrase "numerical control" was first used.

F. By 1955 annual machine tool shows featured tape-controlled machines from as many as seven different machine builders.

G. Today, the numerical controlled machine is an extremely accurate and dependable part of the modern machine shop.

H. Found in many machine shops and manufacturing facilities, the NC machine has a variety of shapes, sizes and uses.

I. This material deals with the uses of NC that are related to the metal removal process--specifically, the vertical and horizontal milling machines and the lathe turning machine.

II. Advantages and disadvantages of numerical control

A. Advantages

1. Greater operator safety
2. Greater efficiency and reliability
3. Less wasted, scrap stock
4. Faster manufacturing times
5. Flexible quantity runs
6. Greater quality control
7. Lower tooling cost
8. Better accuracy and uniformity
9. Lower manpower needs

B. Limitations of numerical control

1. Higher initial investment

2. Higher operating cost

3. Increased electrical maintenance

4. More skilled labor force required

III. Terms and definitions

A. Axis--line along which a major machine tool component travels, such as the table, saddle, spindle or cutting tool

B. Binary-coded decimal--number system which uses combinations of ones and zeros to represent digits in the decimal numbering system

C. CAD (computer-aided design)--design system in which an operator uses a computer for calculation, record keeping, formatting and/or storage

D. CAM (computer-aided manufacturing)--any production system that is computer assisted

E. CNC (computerized numerical control)--type of NC machine that has its own on-board, independent computer which controls the functions and positioning for the specific machine attached to the computer

F. CRT (cathode-ray tube)--type of tube commonly used for computer monitoring screens

G. DNC (direct numerical control)--sometimes referred to as a distributive network of NC machines controlled from a single host computer

H. EIA (Electronic Industries Association)--an electronics trade association which creates technical standards, maintains contact with governmental bodies and distributes marketing data

I. Hardware--structural, mechanical and electrical parts of the NC machine

J. ISO (International Standards Organization)--sets encoding (i.e. disk, punched tape) standards worldwide

K. MCU (machine control unit)--dedicated controlling computer or the panel control system

L. RAM (random access memory)--part of the memory system that can be altered at the MCU and is lost when the controller is turned off

M. ROM (read only memory)--instructions written into the computer's microchip that can be read but not altered

N. Software--the executive program that is resident in the ROM

O. Storage media--agent used for holding and maintaining information for future use, such as floppy disks, punch tape, cassettes, etc.

P. Part program--commands that, when executed, make the NC machine move

CNC ORIENTATION

Match terms on the right with the appropriate definitions.

____ 1.	Reference line from which distances or angles are measured in a coordinate system	a. Axis
____ 2.	Design system in which an operator uses a computer for calculation, record keeping, formatting and/or storage	b. Binary-coded decimal
____ 3.	Commands that make the NC machine move	c. CAD
____ 4.	Type of tube commonly used for computer monitoring screens	d. CAM
____ 5.	Computer memory that is accessible to the part programmer	e. CNC
____ 6.	Computer memory that is written into the electronic chip	ab. CRT
____ 7.	Sometimes referred to as a distributive network of NC machines controlled from a single host computer	ac. DNC
____ 8.	Structural, mechanical and electrical parts of the NC machine	ad. Hardware
____ 9.	Number system which uses combinations of ones and zeros to represent digits in the decimal numbering system	ae. Part program
____ 10.	Type of NC machine that has its own on-board, independent computer	bc. RAM
		bd. ROM
		be. Software

Distinguish between true and false statements by writing an "a" before true statements and a "b" before false ones.

____ 11. Conventional machining is more reliable than NC machining because human judgment does not vary from day to day.

____ 12. NC processes are not efficient.

____ 13. Conventional milling is better suited for producing complex parts.

____ 14. An NC lathe produces a part faster than an engine lathe.

____ 15. For more uniform workpieces, an NC machine should be used.

____ 16. A retrofit milling machine can be used as a hand milling machine.

____ 17. NC machines are not as expensive as conventional tool room machines.

____ 18. Generally, electrical maintenance is high for an NC machine.

____ 19. Conventional machining often has higher tooling costs, as compared to CNC machining.

____ 20. DNC face milling is very often more wasteful than if the job had been done on the shaper.

ANSWERS TO AS 1.1

1. a
2. c
3. ae
4. ab
5. bc
6. bd
7. ac
8. ad
9. b
10. e
11. b
12. b
13. b
14. a
15. a
16. a
17. b
18. a
19. a
20. b

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)

UNIT I: SAFETY GUIDELINES

LESSON 2: HOW NUMERICAL CONTROL WORKS

I. NC system components

A. Numerically coded instructions are classified by format or language. Formatted instructions are merely coordinate points that identify positions along the part's configuration. Language instructions are shortened words that identify and define part geometry and motion statements.

B. The main control unit can read formatted instructions or a programming language and convert this information into a control language by using a post processor.

C. The function of the post processor is to transform the programmed instructions input by the part programmer into very rapid pulses that signal the DC servomotor current off and on. These pulses are recognized as machine movements, i.e. speed, feed, rapid and jog positioning.

D. As long as impulses are on, movement continues. The sensor counts pulses and causes the corresponding turns of the DC servomotors. If the machine is equipped with a closed-loop system, sensors constantly compare impulses with the movement of the servomotor rotation.

E. Early NC machines did not have these pulse/servo-comparators and were known as open-loop systems. They lacked the ability to verify that the machine movements were accurate.

II. Terms and definitions

A. Block--word or group of words that specify a distinct and separate operation

B. Circular interpolation--movement that generates data points along an arc. This data can be contained within one block of information.

C. Closed-loop system--an NC machine system that actively compares the generated output signal to any position the machine moves to

D. Continuous path--operation that maintains a constant rate of movement with no pulses or hesitation between the flow of data

E. Format--organized order that information is required to be in

F. Language--commands specified by text characters used to indicate machine movements

G. Linear interpolation--function that generates data points directly to any positional coordinate without regard for either table axis

H. Open-loop system--machine control system that is unable to compare its generated output signal with the data sent from the MCU

I. Point-to-point--system where endpoints are read by the controller and no new data is read until the movement to that endpoint is achieved

J. Post processor--part of the software which converts cutter-path data into a form that the machine control unit can interpret

K. Servomotor--electronically controlled motor which drives the machine axis

III. NC in the manufacturing plan **(HO 1.1, 1.2)**

A. The engineer or designer creates the blueprint.

B. The NC programmer draws the numerical control sketch.

C. Tooling, i.e. jigs, fixtures and cutting tools, are ordered or made for the job.

D. The planner creates the shop book for efficient routing through the shop. This shop book contains the written record of each stage of the job.

E. The NC programmer develops the written program, the manuscript.

F. Programmed data is loaded into the MCU via punched tape, floppy disks, magnetic tape, DNC, a central host computer, etc.

G. The NC machine is set up with cutters and tooling for the trial run.

H. An NC operator runs the program, diligently monitoring the controls. (Often the trial run is made with expendable or recyclable stock.)

I. The trial-run part is inspected.

J. If necessary, the programmer makes revisions to correct any errors in the data.

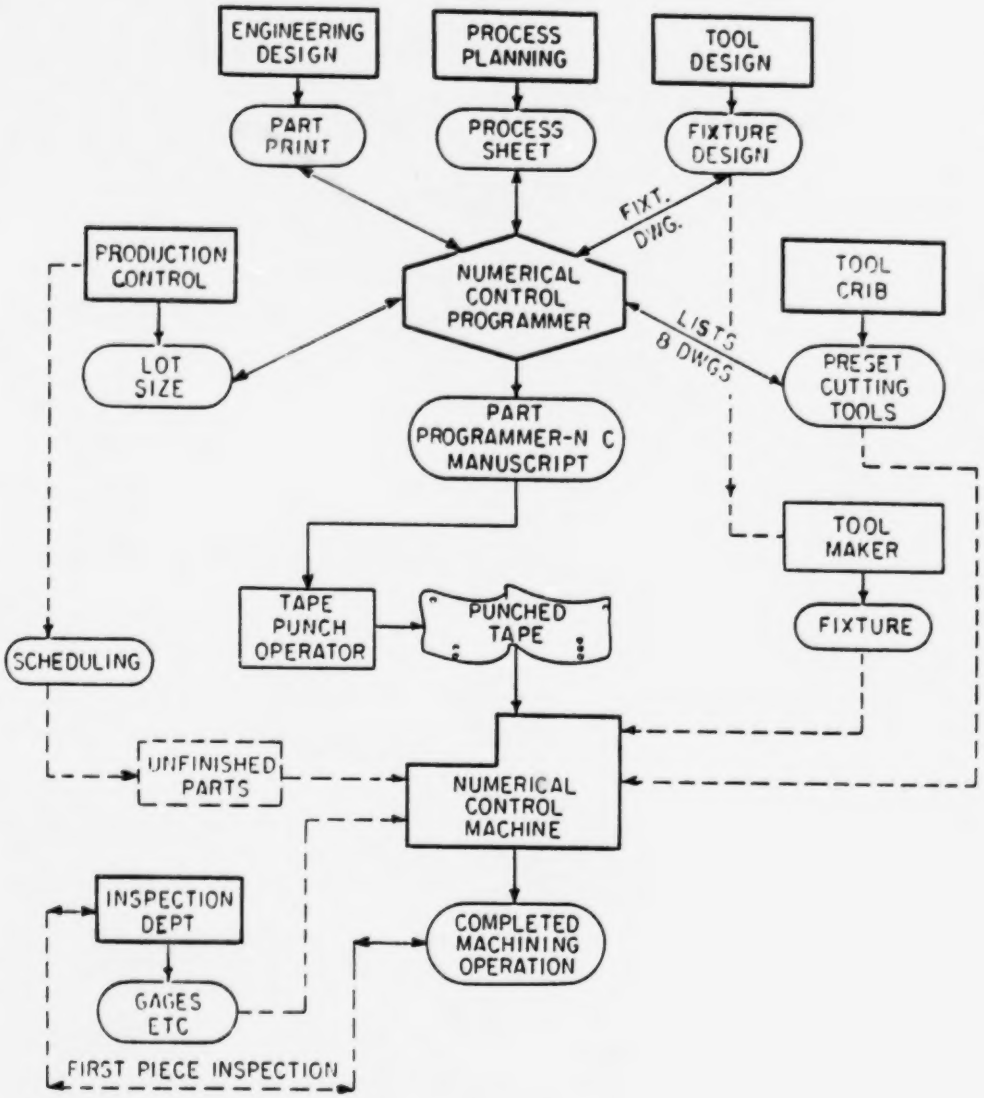
K. Another trial-run part is run.

L. The part is reinspected.

M. If the part passes inspection, the quantity required can now be produced.

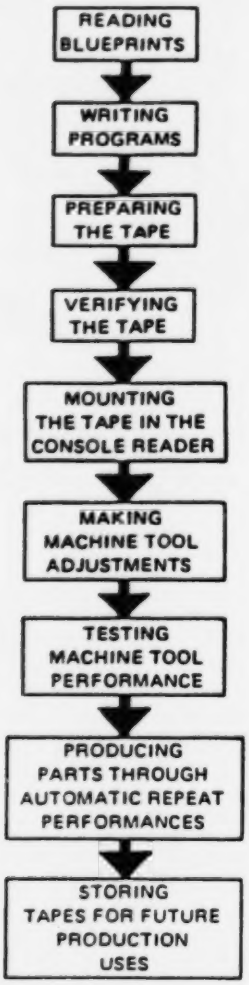
N. The part program can now be stored along with the shop book for future use or reference.

HOW NC FITS INTO THE MANUFACTURING PLAN



Credit: Roberts, Arthur D. Programming for Numerical Control Machines. 2nd ed. New York: McGraw Hill, © 1978, pg. 19. Reproduced with permission.

THE SCOPE OF RESPONSIBILITY (SIMPLIFIED)


READING BLUEPRINTS
WRITING PROGRAMS
PREPARING THE TAPE
VERIFYING THE TAPE
MOUNTING THE TAPE IN THE CONSOLE READER
MAKING MACHINE TOOL ADJUSTMENTS
TESTING MACHINE TOOL PERFORMANCE
PRODUCING PARTS THROUGH AUTOMATIC REPEAT PERFORMANCES
STORING TAPES FOR FUTURE PRODUCTION USES

Credit: Feirer, John L. Machine Tool Metalworking Principles and Practice. 2nd ed. New York: McGraw-Hill, © 1973. Reproduced with permission.

NC PROCEDURES

Read the following carefully, then write "a" before true statements and "b" before false ones.

____ 1. The NC programmer creates the engineering blueprint.

____ 2. Tooling is ordered or made before the NC sketch.

____ 3. The NC machine operator never loads data into the MCU.

____ 4. NC operators are responsible for correcting the programmer.

____ 5. NC data should be stored in a dry, safe place.

____ 6. The NC programmer chooses the tooling required for the job.

____ 7. A program is seldom tested before its production run.

____ 8. Only the operator should make changes to the part program.

____ 9. Once the cycle start-button is pushed, the operator's job is finished.

____ 10. Parts must be thoroughly inspected before full production begins.

Choose the option which best answers the question or completes the statement.

____ 11. An NC language can _____ and _____ part geometry and motion.

a. Receive and list
b. Format and block
c. Define and identify
d. Open and close

____ 12. NC systems must have the _____ to work properly.

a. Programmed language
b. Bits and segments
c. Looping capacity
d. MCU and machine tool

____ 13. An open-loop system _____ .

a. Cannot compare output data
b. Requires programming based on one common datum point
c. Sends signals from the DC servomotor
d. Is part of the program data

___ 14. Numerically coded instructions are classified by _____ or _____ .

a. The processor or sensor
b. Binary digits or signals
c. Language or format
d. Configuration or geometry

Match the terms on the right with the appropriate definition.

___ 15. Movement generated from one block of information along an arc

___ 16. An organized order that information is required to be in

___ 18. Endpoints are read in this system and no new data can be read until movement is achieved.

___ 19. Word or group of words making up a complete, individual command

a. Absolute programming
b. Bit
c. Block
d. Circular interpolation
e. Format
ab. Point-to-point

ANSWERS TO AS 2.1

1. b
2. b
3. a
4. b
5. a
6. a
7. b
8. b
9. b
10. a
11. c
12. d
13. a
14. c
15. a
16. d
17. e
18. ab
19. c

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)

UNIT I: SAFETY GUIDELINES

LESSON 3: SAFETY

I. General safety rules

- A. No horseplay.
- B. Always remove chips from work area with brush or chip rake.
- C. Fasten workpiece to work table securely.
- D. Turn machine off during clean-up procedure.
- E. Always leave the machine clean.
- F. Keep spindle and cutting tool out of the way when loading and unloading stock.
- G. Don't use the control panel top as a coffee counter.
- H. Wear safety glasses in the manufacturing area at all times.
- I. Turn off the spindle whenever measuring the workpiece.
- J. Wear safety shoes.
- K. Use caution when changing tools.
- L. Keep long hair covered or inside a hat while near or operating a machine.
- M. Do not wear jewelry, necktles or loose clothing while operating a machine.
- N. Keep obstacles and tools off the floor.
- O. Lift heavy parts with proper machinery and/or help.
- P. Keep safety guards and devices in place when the machine is in operation.

II. Safety rules specific to numerical control

- A. Do not operate controls unless you have been instructed in their uses.
  1. The cycle stop button stops the machine as soon as the command has been performed.
  2. The emergency stop pushbutton halts the feed of all axes immediately.
- B. Never assume!
- C. Do not attempt to program a machine without programming manuals.
- D. The first line of the program must cancel all modal commands.
- E. Maintain a buffer zone above the workpiece.

F. Do not use compressed air to clean the machine.

G. Do not perform grinding operations near NC machinery.

H. Keep hands away from the rotating spindle and moving turrets.

I. Know where the tool is going at all times.

NC SAFETY

Choose the option which best answers the question or completes the statement.

___ 1. NC programs can be manually loaded in the _____ .

a. Machine
b. Machine control unit
c. Electronic brain
d. Aisle way

___ 2. When measuring the workpiece held down on the table of the machine, _____.

a. The table feed should be on
b. The spindle should be stopped
c. Both of the above
d. None of the above

___ 3. Chips should always be removed from the machine with _____.

a. Your hand
b. An air hose
c. A chip rake or brush
d. A large towel

___ 4. When loading and unloading a workpiece, the operator should _____.

a. Push workpiece near the cutting tool
b. Pry workpiece with a screwdriver
c. Lift workpiece in one steady jerk
d. Retract workpiece a safe distance from cutter

___ 5. Before cleaning the machine, the operator should _____.

a. Turn off master disconnect
b. Take off safety glasses
c. Touch chips to see if they are hot
d. Cover the face

___ 6. Safety glasses should be worn _____.

a. Only if you have a prescription
b. In the shop
c. Only during work shift hours
d. Only around moving objects

___ 7. The control panel of the NC machine _____.

a. Should not be used to hold coffee cups
b. Should always be within reach when cutting the workpiece
c. Should be visible to the operator
d. All of the above

___ 8. Safety shoes _____.

a. Can be substituted for any leather shoe
b. Trap your toes if something heavy falls on them
c. Don't help in the tool room
d. Protect your feet

___ 9. When excessive vibrations or unusual sounds come from the machine, _____.

a. Shut down and investigate
b. Finish the cut, then investigate
c. Call the supervisor but keep the machine running
d. Ignore it--it will go away

___ 10. Which of the following is <u>not</u> a recommended safety procedure?

a. The first line of the program must cancel all model commands.
b. Maintain a buffer zone above the workpiece.
c. Use compressed air to clean the CNC machine.
d. Do not perform grinding operations near NC machinery.

ANSWERS TO AS 3.1

1. b
2. b
3. c
4. d
5. a
6. b
7. d
8. d
9. a
10. c

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)

UNIT I: SAFETY GUIDELINES

UNIT TEST

Choose the option which answers the question or completes the statement.

___ 1. Programmed points, connected by a series of straight-line movements, are referred to as what kind of interpolation?

a. Circular
b. Programmed
c. Lateral
d. Linear
e. Full contouring

___ 2. Which type of interpolation allows the cutting tool to be programmed to move in a circular path ranging from a small arc to a full 360-degree span?

a. Circular
b. Programmed
c. Lateral
d. Linear
e. Curvilinear

___ 3. Which describes a system that uses a computer in the creation or modification of graphic designs?

a. CAD
b. CAM
c. CAN
d. CAAD
e. CAPP

___ 4. What is the main purpose of the post processor?

a. Coordinate programming instruction and specifications
b. Measure a position and convert this measurement into a transmission form
c. Convert cutter-path coordinate data into a form which the machine control can interpret correctly
d. Distinguish positive from negative numbers
e. Establish proper dimensions for the part

___ 5. Which describes a control function where data points are generated between given coordinate positions to allow simultaneous movement of two or more axes of motion in a defined geometric pattern?

a. Interpolation
b. Heuristic
c. Feedback
d. Data processing
e. Incremental

____ 6. How are metal chips and grit accumulations best removed from the machine?

a. Bare hand
b. Gloved hand
c. Brush or rake
d. Direct stream of air
e. Coolant flow

____ 7. What does DNC stand for?

a. Dedicated Numerical Control
b. Digital Number Code
c. Direct Numeric Code
d. Direct Numerical Control
d. None of the above

____ 8. Which is true of the CNC system computer?

a. Capable of processing input information for any CNC machine tool
b. Controls the functions and positioning for the specific machine attached to the computer
c. Requires a host computer to operate
d. Placed at a distance from the machine tool
e. All of the above

____ 9. If the cutter breaks and imposes an immediate safety problem to the operator during the machining of a part, which control function should be activated?

a. Spindle speed override function
b. Emergency stop function
c. Feed hold function
d. Feed rate override function
e. Manual data input

Match the following terms and definitions.

____ 10. Any production system that is computer assisted

____ 11. Type of NC machine that has its own dedicated computer

____ 12. Part of the memory system that can be altered at the MCU and is lost when the controller is turned off

____ 13. Instructions written into the computer's microchip that can be read but not altered

a. Axis
b. Computer-aided design
c. Computer-aided manufacturing
d. Computer-aided data
e. Computerized numerical control
ab. RAM
ac. ROM

Select the option which answers the question or completes the statement.

___ 14. Which button, when depressed, immediately stops the feed of all axes?

a. Cycle stop
b. Emergency stop
c. Spindle stop
d. Axes stop
e. Jog switch

___ 15. How should unsafe work conditions be handled?

a. Ignored by the programmer
b. Reported to the supervisor
c. Corrected by the operator
d. Worked with carefully
e. b and c

___ 16. How should hand tools such as wrenches, measuring equipment, hammers, etc., be managed?

a. Remove from the work area immediately after the machining operations are complete.
b. Remove from the machine and placed on the floor out of the way until needed.
c. Remove from the work area immediately after usage.
d. Place on the machine table out of the tool path.
e. Keep in pockets until needed again.

___ 17. How should cutters and tools be stored?

a. Wherever they fit best
b. With cutting edges down
c. With cutting edges up
d. Wrapped in paper
e. On the machine table

___ 18. To maintain a safe working environment, how often should the CNC machine and work area should be cleaned?

a. Daily
b. Weekly
c. After each use
d. As often as necessary
e. Between shifts only

___ 19. Which is true of safety guards, shields, barriers, covers and protective devices?

a. Must be connected at all times
b. Can be removed to speed machining
c. Must be connected only when operating machine
d. Can be used at the operator's discretion
e. Not necessary in many instances

Match the safety hazard with the appropriate safety equipment.

____	20.	Exposed moving part	a. Safety zones
____	21.	Oil or grease	b. Safety glasses
____	22.	Build-up of chips and grit	c. Guards, shields, barrier covers, etc.
____	23.	Flying metal chips	d. Chip rake or brush
____	24.	Objects falling on floor	e. Safety shoes
____	25.	Sharp edges	ab. Gloves
			ac. Degreasing agent
			ad. Ear plugs

NOTE: Completion of this test to specified score is the student's final evaluation for competency M1: Employ CNC machine safety guidelines consistently.

ANSWERS TO UNIT TEST

1.	d	
2.	a	
3.	a	
4.	c	
5.	a	
6.	c	(M1)
7.	d	
8.	b	
9.	b	
10.	c	
11.	e	
12.	ab	
13.	ad	
14.	b	(M1)
15.	e	(M1)
16.	c	
17.	b	
18.	d	(M1)
19.	c	(M1)
20.	c	(M1)
21.	ac	(M1)
22.	d	(M1)
23.	b	(M1)
24.	e	(M1)
25.	ab	(M1)

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)
UNIT II: COORDINATES AND DIMENSIONS

UNIT OBJECTIVE

Upon completion of this unit, the student should demonstrate understanding of the Cartesian coordinate system, describe a conventionally dimensioned workpiece via the numerical control (NC) drawing, describe the binary-coded decimal system, and identify formats and languages used in NC. This knowledge will be evidenced by achieving the specified score on assignment sheets and the unit test.

SPECIFIC OBJECTIVES

After completion of the lessons in this unit, the student should be able to:

1.1 Calculate coordinates and dimensions for a CNC drawing.

1.2 Identify basic axis movements.

1.3 Explain NC drawing nomenclature.

2.1 Describe the binary-coded decimal system.

3.1 Match basic tab sequential format terms with definitions. (competency M2)

3.2 Identify word address format uses.

3.3 Identify languages available for NC. (competency M4)

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)

UNIT II: COORDINATES AND DIMENSIONS

CONTENTS OF THIS UNIT

A. Objective sheet

B. References

C. Notes to the instructor

D. Lesson plans

1. Lesson 1: COORDINATE SYSTEM

   a. Information outline

   b. Handouts

      HO 1.1: Cartesian coordinate system
      HO 1.2: The axes
      HO 1.3: Axis travel for milling machines
      HO 1.4: Axis travel for the lathe

   c. Assignment sheets

      AS 1.1: NC drawings
      AS 1.2: Cartesian coordinate practice

   d. Answers to assignment sheets

2. Lesson 2: BINARY-CODED SYSTEM

   a. Information outline

   b. Assignment sheet

      AS 2.1: System comparisons

   c. Answers to assignment sheet

3. Lesson 3: NUMERICAL CONTROL FORMATS

   a. Information outline

   b. Assignment sheet

      AS 3.1: Tab sequential format

   c. Answers to assignment sheet

E. Unit test

F. Answers to unit test

REFERENCES

A. Books

1. Luggen, William W. Fundamentals of Numerical Control. 2nd ed. Albany, NY: Delmar Publishers, Inc., 1988.

2. Roberts, Arthur D. Programming for Numerical Control Machines. 2nd ed. New York: McGraw Hill, 1978.

3. Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar Publishers, Inc., 1986.

4. Stranton, George C. Numerical Control Programming. 1st ed. New York: John Wiley & Sons, Inc., 1988.

B. Others

1. A Catalog of Performance Objectives and Performance Guides for Computerized Numerical Control. Atlanta, GA: Vocational-Technical Education Consortium of States; Carbondale, IL: Department of Vocational Education, Southern Illinois University, June 1985.

2. Computerized Numerical Control (test item bank). Atlanta, GA: Vocational-Technical Education Consortium of States; Carbondale, IL: Dept. of Vocational Education Studies, Southern Illinois University, 1986.

3. Theories and Operation of Numerical Control/Computerized Numerical Control (NC/CNC) (U105001). Urbana: University of Illinois at Urbana-Champaign: Vocational Education Service, March 1988.

NOTES TO THE INSTRUCTOR

A. Suggested approaches to teaching this unit

1. Discuss the size, shape, parity check requirements and channels necessary to express data via the EIA standard 1" wide tape.

2. Appraise each student's ability to concentrate on details and cope with frustration.

3. Place emphasis on independent thinking and sound logic.

4. Demonstrate drawing and dimensioning procedures.

5. Sketch an angle plate on the board using conventional drafting language.

B. Suggested student activities

1. Require students to create an NC sketch of an angle plate.

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)

UNIT II: COORDINATES AND DIMENSIONS

LESSON 1: COORDINATE SYSTEM

I. The numerical control (NC) drawing

A. A major responsibility of NC programming is to interpret the blueprint correctly.

B. Interpreting the blueprint is a skill which can be developed with practice. Nevertheless, the student must know and understand basic blueprint conventions.

C. The student will be called upon to convert standard blueprints into NC drawings.

D. The NC drawing is an accurate, to-scale and fully developed document.

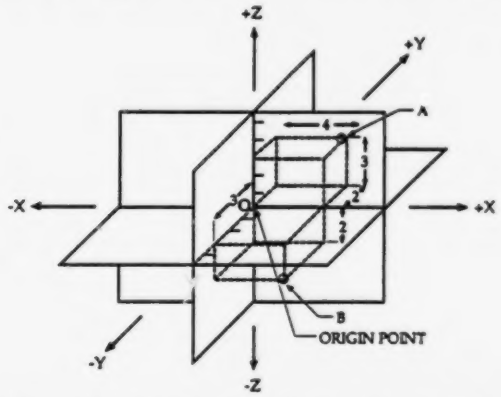
E. NC drawings are based on the principle of polar and rectangular coordinate positioning.

1. Polar coordinates are located by extending the radius line from the origin point to the hole's center, then measuring the angular distance counter-clockwise from the X axis.

2. Positive polar coordinates identify a counter-clockwise position from the X axis, while negative polar coordinates identify a clockwise position from the X axis.

F. Rectangular coordinate positioning, absolute, is better known as the Cartesian coordinate system. **(HO 1.1)**

1. This system of defining mathematical points in a three-dimensional model was discovered by the French philosopher and mathematician René Descartes more than 300 years ago.



Credit: Theories and Operation of Numerical Control/Computerized Numerical Control (NC/CNC) (U105001). Urbana: University of Illinois at Urbana-Champaign: Vocational Education Service, March 1988, pg. 19. Developed under contract with the Illinois State Board of Education.

2. The right-hand rule applies to the rectangular coordinate system.

3. The Cartesian coordinate system is divided into four quadrants **(HO 1.1)**.

a. Each quadrant is numbered beginning with the upper right-hand section, counting counter-clockwise.

b. Lines that divide or separate quadrants are called axes.

4. For clarification, basic axes are designated by letters from the alphabet: X, Y, Z, A, B, C.

G. The incremental system locates points in terms of distance and direction from the previous location, not origin.

II. Axis nomenclature applied to NC machines (HO 1.2-1.4)

A. The Cartesian coordinate system is overlaid on a milling machine's table and spindle and on a lathe's table and cross-slide.

B. Vertical milling machine

1. X-axis movement causes the table to travel left and right as the operator faces the machine from the front.

2. Y-axis movement is away from and toward the operator as the operator faces the machine from the front.

C. Horizontal milling machine

1. X-axis movement is left and right.

2. Y-axis movement is up and down.

D. Lathe

1. X-axis movement is parallel to the cross-slide.

2. Z-axis movement of lathes and milling machines is always parallel to the spindle.

III. NC drawing nomenclature and terms

A. Chamfer--to cut off an edge or corner; flat surface resulting when a sharp corner is cut off

B. Flange--angular or curved contour of a part's outer surface

C. Isometric projection--view of an object where all three faces are equally inclined to the drawing surface; all edges are equally foreshortened

D. Kerf--groove or notch made in stock by a saw or other cutting tool

E. Neck--to cut a circular groove around a shaft

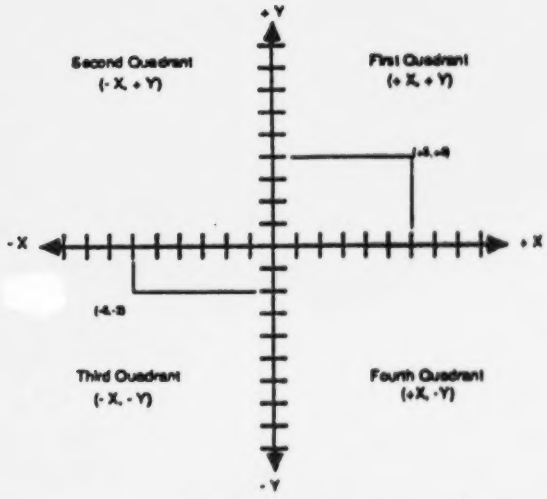
F. Orthographic projection--view of an object so that two of its axes make equal angles with the plane of projection (usually three separate views--top, front and end)

G. Perspective--view of an object using one plane; object is positioned so that three faces are displayed

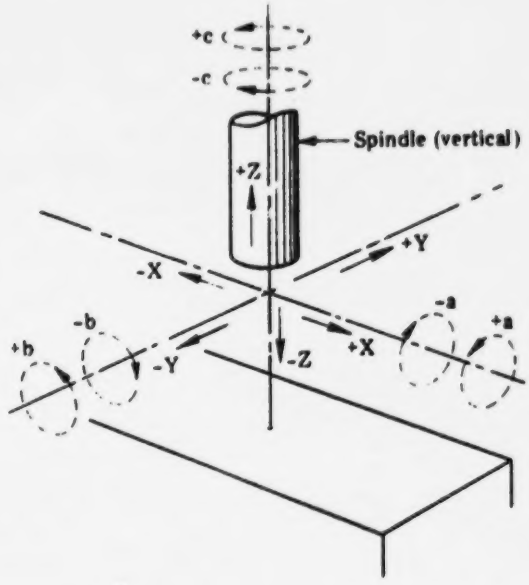
H. Plotter--device that draws, plots or traces out NC data

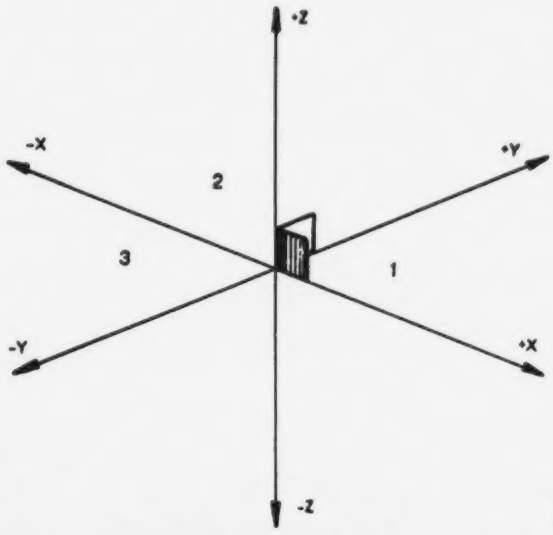
I. Roadmap--drawing which describes the path of the cutting tool as it progresses around a part

CARTESIAN COORDINATE SYSTEM

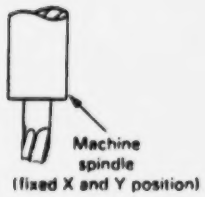


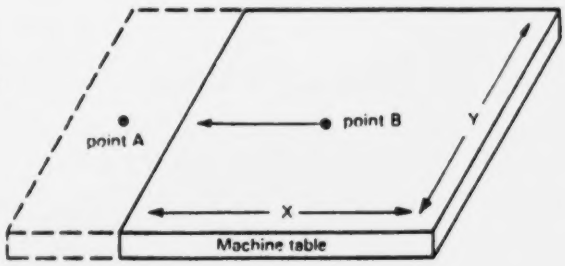
TWO-AXIS COORDINATE SYSTEM





THREE-AXIS COORDINATE SYSTEM

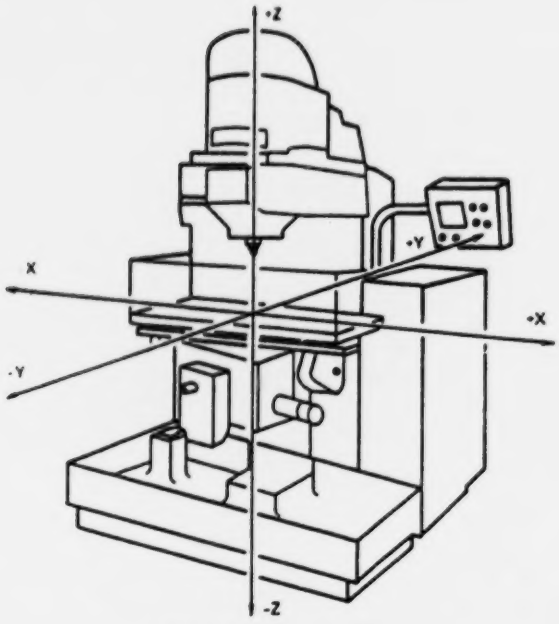




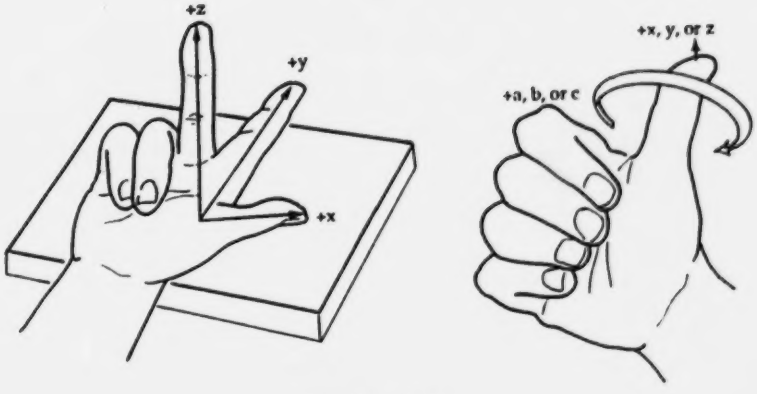
Credits: Olivo, C. Thomas. Advanced Machine Tool Technology. North Scituate, MA: Breton Publishers, a division of Wadsworth, Inc., 1982, pg. 645.

Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar Publishers, Inc., © 1986. Reproduced by permission.

THE AXES



Three-axis vertical mill

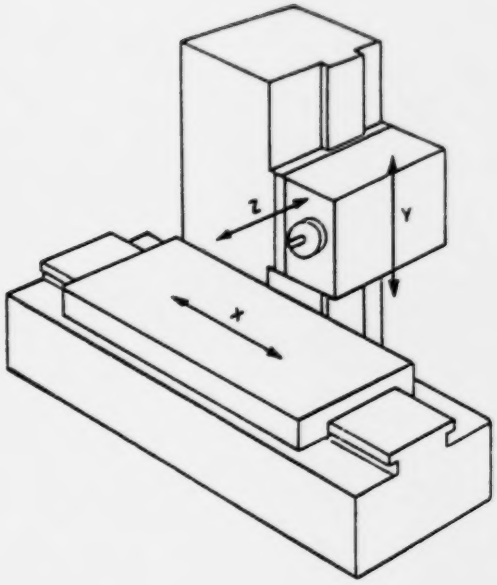


Right-hand rule

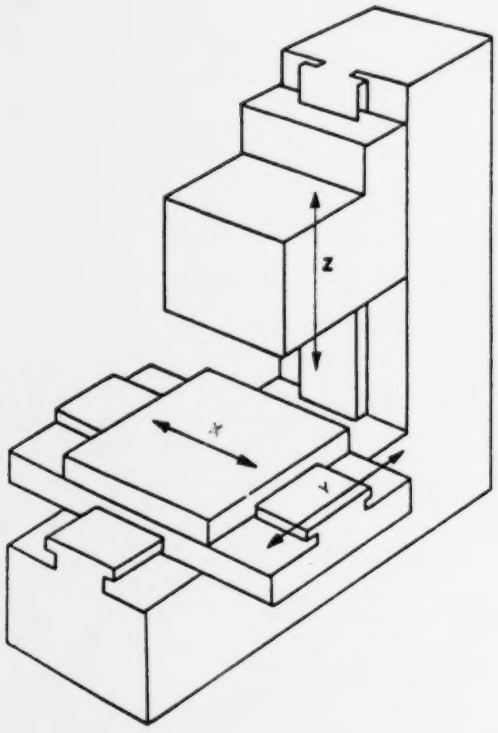
Credits: Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar Publishers, Inc., © 1986, pg. 27. Reproduced by permission.

Theories and Operation of Numerical Control/Computerized Numerical Control (NC/CNC) (U105001). Urbana: University of Illinois at Urbana-Champaign: Vocational Education Service, March 1988. Developed under contract with the Illinois State Board of Education.

AXIS TRAVEL FOR MILLING MACHINES



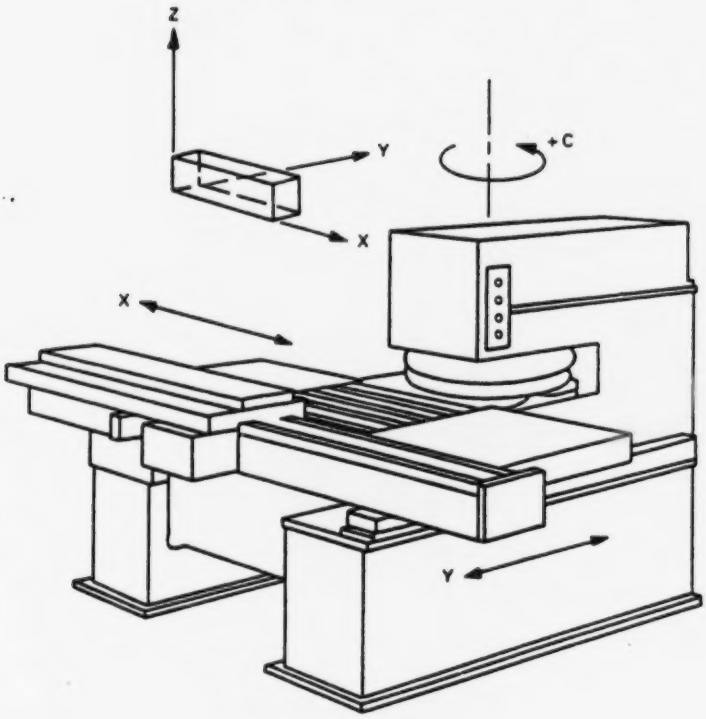
A horizontal N C machine



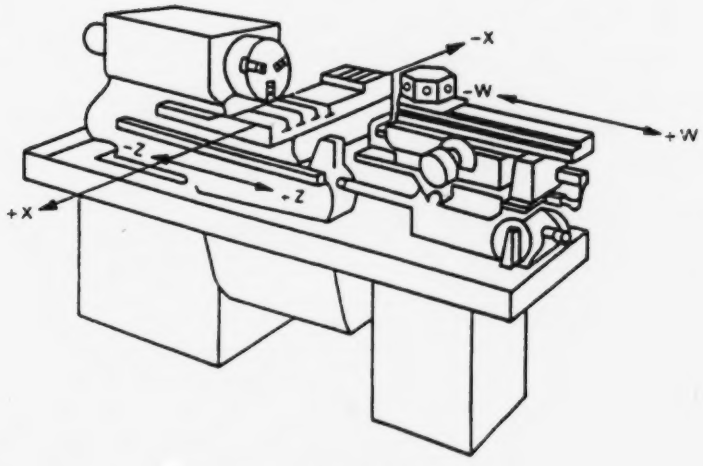
A vertical N C machine

Credit: Luggen, William W. Fundamentals of Numerical Control. 2nd ed. Albany, NY: Delmar Publishers, Inc., © 1988, pg. 29. Reproduced by permission.

AXIS TRAVEL FOR THE LATHE



N C turret punch press



N C turret lathe

Credit: Luggen, William W. Fundamentals of Numerical Control. 2nd ed. Albany, NY: Delmar, © 1988, pg. 30. Reproduced by permission.

NC DRAWINGS

Write an "a" before true statements and a "b" before false ones.

____ 1. A standard blueprint should not be used as the source for the NC drawing.

____ 2. The NC drawing is the same as the NC sketch.

____ 3. NC drawings sometimes use polar coordinates for hole patterns.

____ 4. Polar coordinate and Cartesian coordinate systems are the same.

____ 5. When a polar coordinate is designated as "ccw", it is understood to be positive.

____ 6. The right-hand rule always points the thumb in the -X direction.

____ 7. Quadrant 3 can be described simply as "+X, -Y".

____ 8. The polar coordinate system and the rectangular coordinate system are <u>not</u> one and the same.

____ 9. The middle finger in the right-hand rule designates the Z axis.

____ 10. An NC sketch does not have to be as precise as the blueprint.

Match the terms on the right with the appropriate definitions.

____ 11. Up-and-down spindle movement on a vertical milling machine

____ 12. Left-and-right movement of the horizontal milling machine's table

____ 13. Table travel movement on the vertical milling machine

____ 14. In-and-out movement on the saddle of the vertical mill

____ 15. In-and-out movement of the lathe's cross-slide

____ 16. Movement parallel to the ways

____ 17. In-and-out spindle movement on the horizontal milling machine

____ 18. Circular movement around the Z axis

____ 19. Circular movement around the Y axis

____ 20. Circular movement around the X axis

a. A axis

b. B axis

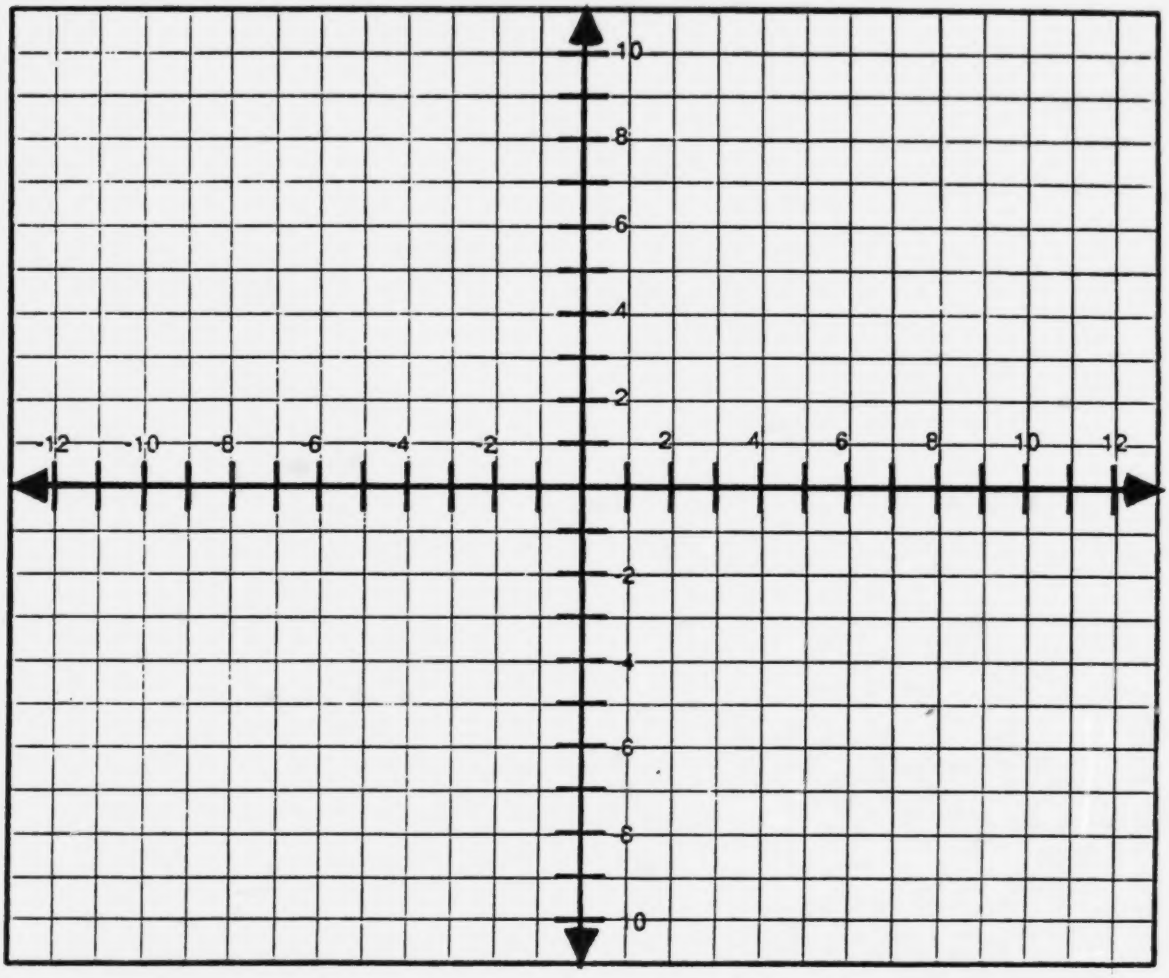
c. C axis

d. X axis

e. Y axis

ab. Z axis

ABSOLUTE AND INCREMENTAL PRACTICE



Using the absolute mode, plot the points below on the graph. Label each point with the letter that appears before it.

A. (+3, -4)
B. (-5, -6)
C. (+8, +7)
D. (-1, -3)
E. (-4, +3)
F. (+10, -8)

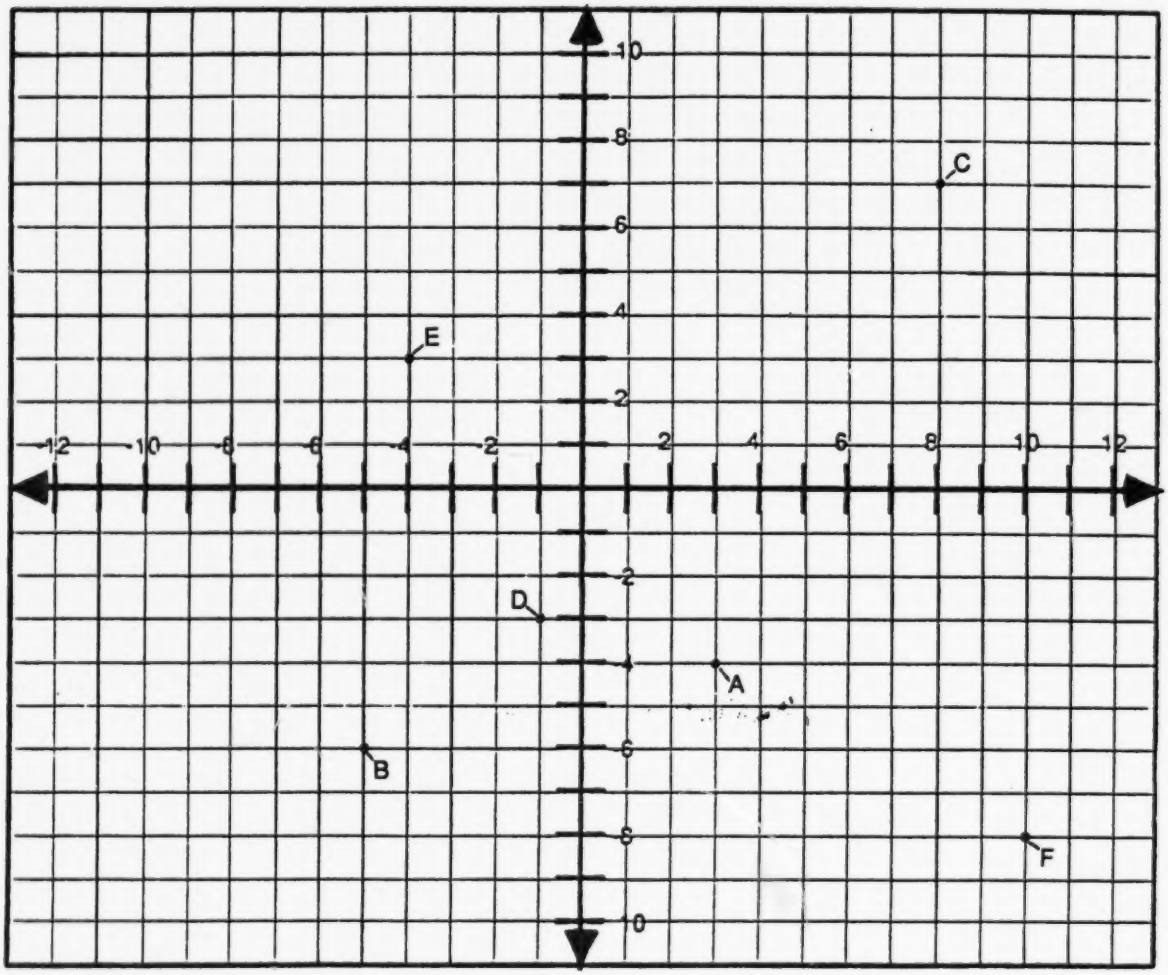
Using the incremental mode, identify each of the previous positions. (Example: A = X +3, Y -4; B = X -8, Y -2)

1. C = X _____, Y _____
2. D = X _____, Y _____
3. E = X _____, Y _____
4. F = X _____, Y _____

ANSWERS TO AS 1.1

1. b
2. a
3. a
4. b
5. a
6. b
7. b
8. a
9. a
10. b
11. ab
12. d
13. d
14. e
15. d
16. ab
17. ab
18. c
19. b
20. a

ANSWERS TO AS 1.2



1. +13, +13
2. -8, -10
3. -3, +6
4. +14, -11

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT II: COORDINATES AND DIMENSIONS

LESSON 2: BINARY-CODED SYSTEM

I. Binary code in numerical control

A. Historically, the most common way to program the NC machine has been with a binary-coded punched tape.

NOTE: The major difference between NC and CNC is that CNC does not program the machine with punched tape.

1. Punched tape stores information by using the binary system of counting.

2. Binary code is based on the mathematical system structured on the power of two.

3. The Base 2 system (binary) can be understood easily by comparing it to the more familiar Base 10 (Arabic) system.

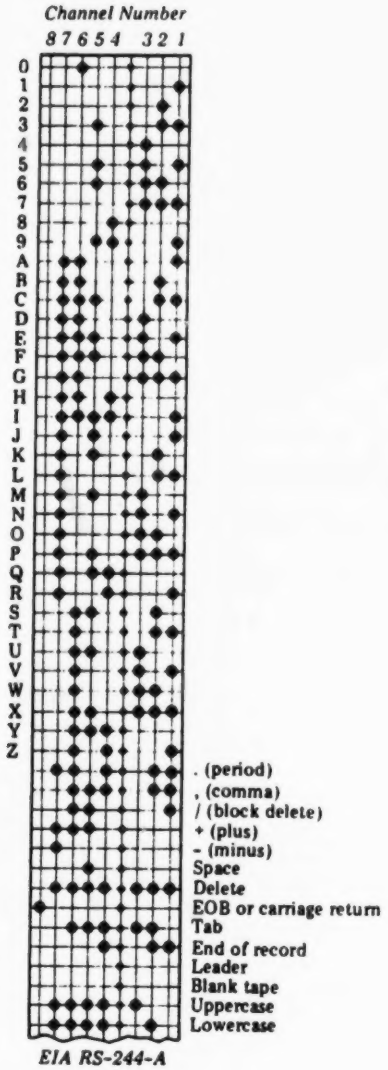
4. Base 2 counts from 0 to 1; the only numbers used are 0 and 1.

Arabic	Binary	Arabic	Binary
0	0	18	10010
1	1	19	10011
2	10	20	10100
3	11	21	10101
4	100	22	10110
5	101	23	10111
6	110	24	11000
7	111	25	11001
8	1000	26	11010
9	1001	27	11011
10	1010	28	11100
11	1011	29	11101
12	1100	30	11110
13	1101	31	11111
14	1110	32	100000
15	1111	64	1000000
16	10000	128	10000000
17	10001		

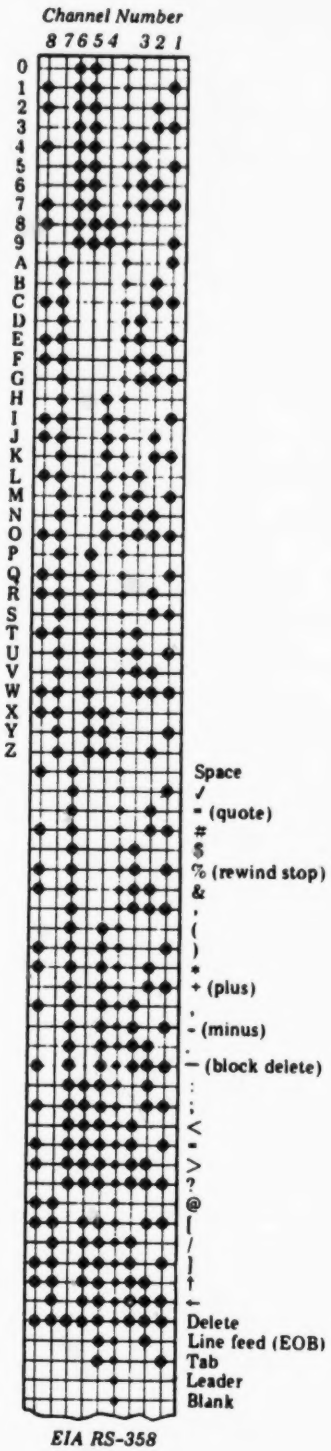
B. Two standards have been developed so that letters, Base 10 numbers and other characters can be converted into binary numbers.

1. RS-244 is the standard for the Binary Coded Decimal system (BCD).

2. RS-358 is the standard for the American Standard Code for Information Interchange (ASCII).


Channel Number
8 7 6 5 4 3 2 1
0 1 2 3 4 5 6 7 8 9 A B C D E F G H I J K L M N O P Q R S T U V W X Y Z
. (period)
, (comma)
/ (block delete)
+ (plus)
- (minus)
Space
Delete
EOB or carriage return
Tab
End of record
Leader
Blank tape
Uppercase
Lowercase

EIA RS-244-A format



EIA RS-358 format

Credit: Olivo, C. Thomas. Advanced Machine Tool Technology. North Scituate, MA: Breton Publishers, a division of Wadsworth, Inc., 1982, pg. 648.

SYSTEM COMPARISONS

Write an "a" before true statements and a "b" before false ones.

____ 1. There is no way to compare the Arabic and binary systems of numbering.

____ 2. Binary-coded punched tape is an easy, inexpensive way to store data.

____ 3. Computerized Numerical Control uses the punched tape for programming.

____ 4. Binary code is based on a mathematical system using the numbers 1 and 2.

____ 5. Like Base 10, the binary system does not convert alpha characters.

____ 6. The number 0 is used by RS-244 and RS-358 standards.

____ 7. Binary code can only be stored on punched tape.

____ 8. Base 10 begins with the number 1 and counts through to 10.

ANSWERS TO AS 2.1

1. b
2. a
3. b
4. b
5. b
6. a
7. b
8. b

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT II: COORDINATES AND DIMENSIONS

LESSON 3: NUMERICAL CONTROL FORMATS

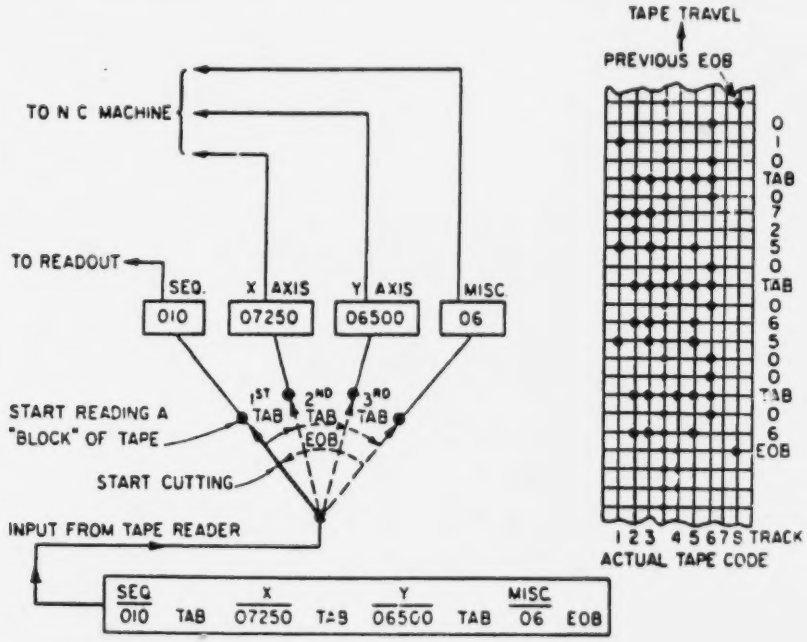
I. The tab sequential format

A. The tab code is a wall between signals generated by the binary punch tape being read into the MCU.

B. The tab functions as a stepping-switch that designates the separation of one complete bit of information from another.

C. Thus the tab "steps" to each bit of input data in order to sharply define information punched on the coded tape.

D. This segmenting operation establishes the data in a readable format.



Credit: Roberts, Arthur D. Programming for Numerical Control Machines. 2nd ed. New York: McGraw Hill, © 1978. Reproduced with permission.

E. From the illustration above you can see that the sequence of operation is the first data to be input, then comes a tab.

F. Next is some dimension relative to the X axis followed by another tab.

G. Y-axis data is now input after which comes another tab.

H. Now comes some bit of miscellaneous information followed by the EOB command. The EOB (end-of-block) command resets the stepping-switch so that the MCU can receive the signal designating a new sequence number.

II. The word address format

A. Word address, letter address or variable block are different names for the same format.

B. All refer to the format using program lines (blocks) of variable length.

N_ _ _G_ _X_ _._ _ _Y_ _._ _ _Z_ _._ _ _R_ _._ _F_ _ _._S_ _T_ _M_ _H_ _P_ _Q_ _

1. N = sequence number; designates the start of a reference block. On some machinery, an "O" may start a block of information.

2. G = preparatory function code which changes the control mode of the machine (drilling, tapping, boring, etc.)

3. X = X-axis coordinate; also used to enter time intervals

4. Y = Y-axis coordinate

5. Z = Z-axis coordinate

6. R = arc radius

7. F = feed rate

8. S = spindle speed (rpm)

9. T = tool to be used in a tool change

10. M = miscellaneous functions such as spindle and coolant control or end of a program

11. H = offset code

12. P = sub-program code

13. Q = canned cycle parameter

C. Preceding each bit of data is a letter. This letter is a code that acts as the stepping-switch in order to separate program data.

III. NC computer languages

A. The increased use of computers in NC has overcome many of the limitations of the binary coded tape.

B. More complex commands can be made with computers using a language than with formats.

C. Computers can save hundreds of work hours by processing data more quickly and more accurately than punched tape.

D. With the increase of computers in the manufacturing fields came many cost reducing aids.

E. Dozens of NC processor languages were created in order to help diagnose human error before it reached the machine shop floor.

F. It is easier to detect and remedy programming errors at the programming station, rather than at the machine tool.

G. Computer programming languages range from conversationally simple to extremely complex.

H. What distinguishes one language from another is the quantity of vocabulary words, their explicit instruction arrangement and the size of the computer required.

I. Types of NC languages

1. APT (Automatically Programmed Tools) was the first language developed for NC.

   a. APT also has the largest vocabulary of the general processor languages.

   b. Because the APT language has the most extensive vocabulary, a large-capacity computer is required to use it.

2. Because of cost constraints, a less extensive vocabulary language was needed. AD-APT (Air Material Command-Developed APT) was developed for medium-sized computers.

3. AUTOMAP (AUTOmatic MAchining Program) was a further outgrowth of APT that was significantly less complicated and easy to learn.

4. COMPACT II is a very flexible and easy to learn language that can operate on the computer time-sharing basis, thereby eliminating the need for individual shops to purchase their own mainframe computer.

5. UNIAPT uses a batch mode on a local dedicated minicomputer.

6. NVFORM language uses an 80-column form onto which the programmer places the appropriate command data.

TAB SEQUENTIAL FORMAT

Match the terms on the right with the appropriate definitions.

___ 1. The first data on each line of a formatted command is the _____ .

___ 2. Each program line of complete information is a(n) _____ .

___ 3. The _____ code forms a wall between input data.

___ 4. The _____ carries the entire program to be read by the MCU.

___ 5. In the block format for word address the "T" specifies the _____ .

___ 6. The tab functions as a(n) _____ .

___ 7. The "S" in word address block format represents _____ .

___ 8. The _____ initiates a preparatory function.

___ 9. The _____ controls auxiliary functions.

___ 10. At the end of every program line is a(n) _____ designator.

a. Block
b. Binary punch tape
c. EOB
d. Feed rate
e. G code
ab. M function
ac. Sequence number
ad. Spindle rpm
ae. Stepping switch
bc. Tab
bd. Tool

ANSWERS TO AS 3.1

1. ac
2. a
3. bc
4. b
5. bd
6. ae
7. ad
8. e
9. ab
10. c

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT II: COORDINATES AND DIMENSIONS

UNIT TEST

Select the option which answers the question or completes the statement.

____ 1.	Angular or curved contour of a part's outer surface	a.	Chamfer
____ 2.	To cut off an edge or corner; flat surface resulting when a sharp corner is cut off	b.	Flange
____ 3.	To cut a circular groove around a shaft	c.	Isometric
____ 4.	Code which forms a wall between input data	d.	Neck
____ 5.	View of an object using one plane; object is positioned so that three faces are displayed	e.	Orthographic
____ 6.	View of an object where all three faces are equally inclined to the drawing surface	ab.	Perspective
____ 7.	View of an object so that two of its axes make equal angles with the plane of projection	ac.	Plotter
		ad.	Tab

Select the option which answers the question or completes the statement.

____ 8. What term refers to expressing a dimension with respect to the origin of a coordinate system?

a. Accumulated
b. Incremental
c. Proportional
d. Absolute
e. None of the above

____ 9. Which coordinate system defines the position of the point, line or plane in terms of a radius and the angle between the positive X-axis and the geometric shape?

a. Polar
b. Absolute
c. Precision
d. Cartesian
e. None of the above

____ 10. What type of drawing describes the path of the cutting tool as it progresses around a part?

a. Read-out
b. Track
c. Road map
d. Manuscript
e. Blueprint

___ 11. What device will draw, plot or trace from coded NC data input?

a. Tracer
b. Interpolator
c. Plotter
d. Invertor
e. None of the above

___ 12. What does an orthographic projection show?

a. A pictorial representation of the part
b. An exploded view of the part assembly
c. How the part fits into the work-holding device
d. Three views of the part
e. None of the above

___ 13. Which axis is horizontal and parallel to the work-holding surface?

a. Z
b. C
c. Y
d. X
e. W

___ 14. When positioning the workpiece in relation to the machine axis, the depth of the part usually corresponds to which axis?

a. Z
b. Y
c. X
d. W
e. C

___ 15. Which axis shows rotary motion around the X-axis?

a. X
b. C
c. A
d. H
e. W

___ 16. Which of the following is true of an NC drawing?

a. Accurate and to-scale
b. Based on polar and rectangular coordinate positioning
c. Uses the Cartesian coordinate system frequently
d. Created from standard blueprints
e. All of the above

___ 17. What is the major difference between NC and CNC?

a. NC uses punched tape and the Base 10 system of counting
b. CNC uses the Base 2 system, while NC uses Base 10
c. CNC uses an on-board, independent computer
d. NC does not program the machine with punched tape
e. None of the above

___ 18. Which is binary for the Arabic numeral 12?

a. 0100
b. 1100
c. 1110
d. 1111
e. 222

___ 19. What is a block within a CNC program identified by an "O" or "H" in place of the word address "N" called? (It contains sufficient data to enable resumption of the program following an interruption.)

a. Scanner
b. Repeat
c. Reference
d. Transfer
e. None of the above

___ 20. Which was developed as a good, affordable general processor language?

a. APT
b. AD-APT
c. COMPACT II
d. NVFORM
e. CAD

___ 21. Which language is flexible, easy to learn and can be operated on a time-sharing basis?

a. APT
b. AD-APT
c. COMPACT II
d. NVFORM
e. CAD

Match the following tape format definitions with terms.

___ 22. Symbol indicating the significance of the information immediately following

___ 23. Word or group of words, considered as units and providing sufficient information for an operation

___ 24. Paths parallel to the edge of the tape along which information can be stored

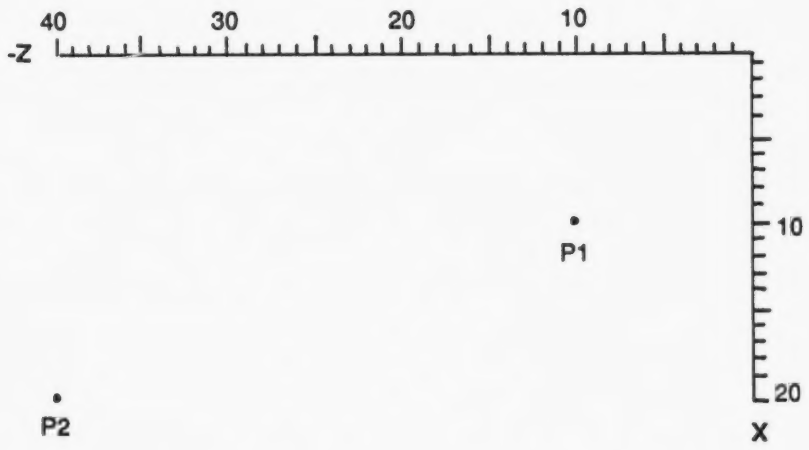
___ 25. Binary digit represented by the presence or absence of a hole on the tape

___ 26. Coded representation of symbols such as letters, numerals and punctuation marks on the tape

___ 27. Path perpendicular to the edge of the tape along which information can be stored

a. Address
b. Bit
c. Block
d. Channels
e. Characters
ab. Offset
ac. Row

Using the graph below, express lathe movement in absolute and incremental modes.



____ 28. In absolute, describe point 2.

a. X 20, Z -40
b. X 20, Z -30
c. X 10, Z -20
d. X 10, Z -10
e. X -10, Z 30

____ 29. Using incremental, moving to point 2 from point 1 would be _____.

a. X 20, Z -40
b. X 20, Z -30
c. X 10, Z -20
d. X 10, Z -10
e. X -10, Z 30

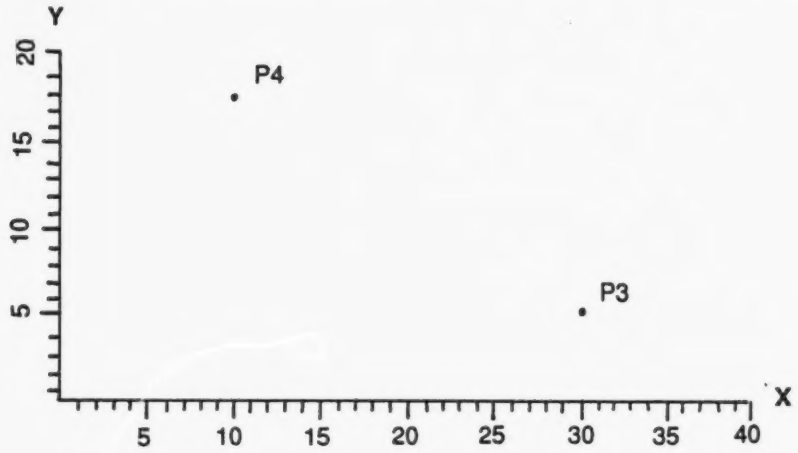
____ 30. In absolute, describe point 1.

a. X 10, Z 10
b. X 10, Z -30
c. X 10, Z -10
d. X -10, Z 10
e. X -10, Z 20

____ 31. Using incremental, moving to point 1 from point 2 would be _____.

a. X 10, Z 10
b. X 10, Z -30
c. X 10, Z -10
d. X -10, Z 10
e. X -10, Z 20

Using the graph below, express milling machine movement in absolute and incremental modes.



___ 32. In absolute, point 4

a. X 10, Y -18
b. X -20, Y -13
c. X 20, Y -18
d. X 10, Y 18
e. X -20, Y 13

___ 33. Using incremental, moving to point 4 from point 3

a. X 10, Y -18
b. X -20, Y -13
c. X 20, Y -18
d. X 10, Y 18
e. X -20, Y 13

___ 34. In absolute, point 3

a. X 20, Y -13
b. X -20, Y -13
c. X -30, Y 5
d. X 30, Y 5
e. X 30, Y -5

___ 35. Using incremental, point 3 from point 4

a. X 20, Y -13
b. X -20, Y -13
c. X -30, Y 5
d. X 30, Y 5
e. X 30, Y -5

NOTE: Completion of this test to specified score is the student's final evaluation for competency M3: Calculate coordinates and dimensions of CNC drawing, and partial evaluation for M4: Write program for CNC machine.

ANSWERS TO UNIT TEST

1.	b	
2.	a	
3.	d	
4.	ad	
5.	ab	
6.	c	
7.	e	
8.	d	(M3)
9.	a	(M3)
10.	c	
11.	c	
12.	d	
13.	d	(M3)
14.	a	(M3)
15.	c	(M3)
16.	e	
17.	c	
18.	b	
19.	c	(M4)
20.	b	(M4)
21.	c	(M4)
22.	a	(M4)
23.	c	(M4)
24.	d	
25.	b	
26.	e	
27.	ac	
28.	a	(M3)
29.	e	(M3)
30.	c	(M3)
31.	b	(M3)
32.	d	(M3)
33.	e	(M3)
34.	c	(M3)
35.	a	(M3)

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)
UNIT III: NC MATH

UNIT OBJECTIVE

Upon completion of this unit, the student should understand cutter center-line offsets and NC math for programming. This knowledge will be evidenced by achieving the specified score on assignment sheet and the unit test.

SPECIFIC OBJECTIVES

After completion of the lessons in this unit, the student should be able to:

1.1 Determine tool positioning using center-line (C-L) data.

1.2 Solve problems concerning simple trigonometry. (partial competency M4)

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)

UNIT III: NC MATH

CONTENTS OF THIS UNIT

A. Objective sheet

B. References

C. Notes to the instructor

D. Lesson plans

1. Lesson 1: BASIC TRIGONOMETRY FOR NC

   a. Information outline

   b. Transparencies

      TM 1.1: Understanding cutter center-line offsets
      TM 1.2: Understanding cutter center-line offsets (cont.)

   c. Handouts

      HO 1.1: Right triangles
      HO 1.2: Oblique triangles
      HO 1.3: Center-line offsets

   d. Assignment sheet

      AS 1.1: Simple trigonometry

   e. Answers to assignment sheet

E. Unit test

F. Answers to unit test

REFERENCES

A. Books

1. Luggen, William W. Fundamentals of Numerical Control. 2nd ed. Albany, NY: Delmar Publishers, Inc., 1988.

2. Roberts, Arthur D. Programming for Numerical Control Machines. 2nd ed. New York: McGraw Hill, 1978.

3. Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar Publishers, Inc., 1986.

4. Stranton, George C. Numerical Control Programming. 1st ed. New York: John Wiley & Sons, Inc., 1988.

B. Others

1. Computerized Numerical Control (test item bank). Atlanta, GA: Vocational-Technical Education Consortium of States; Carbondale, IL: Dept. of Vocational Education Studies, Southern Illinois University, 1986.

NOTES TO THE INSTRUCTOR

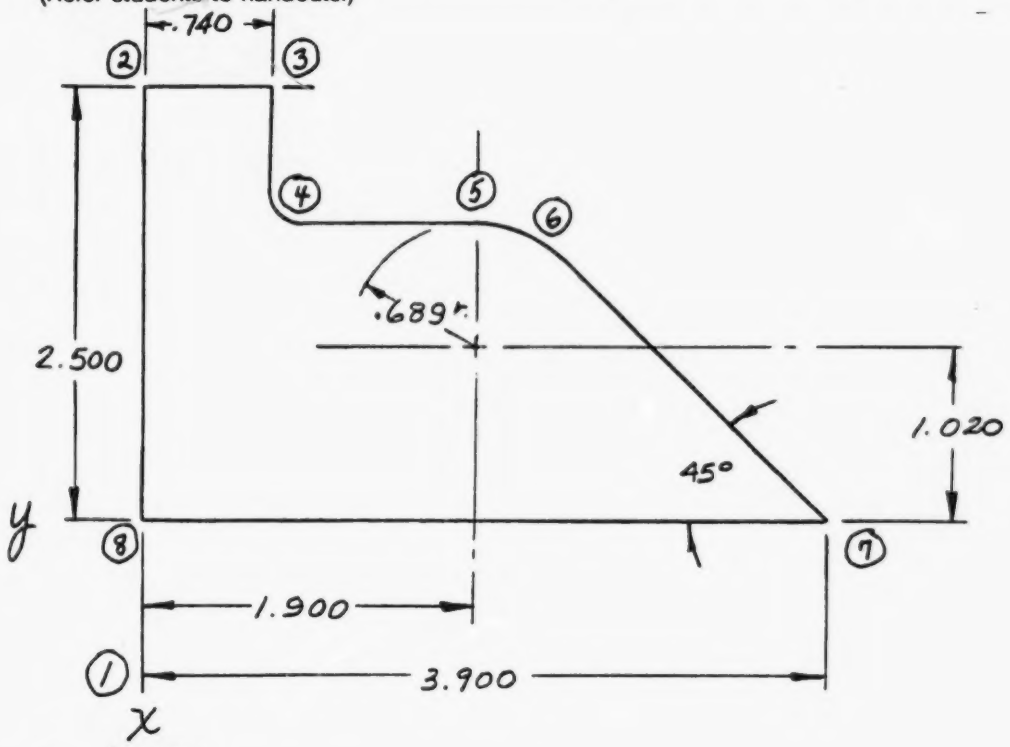
A. Approaches to teaching this unit

1. Stress the absolute need for understanding right-angle trigonometry.

2. Assess each student's ability to concentrate on details and cope with frustration.

3. Place emphasis on independent thinking and sound logic.

4. Create trigonometry problems related to cutter center-line on the board. For example, using a 3/8" diameter end mill, determine the coordinate position for the following center-line offset. (Refer students to handouts.)



a. Answers

1) X -.1875, Y -1.00

2) X -.1875, Y 2.6875

3) X .9275, Y 2.6875

4) X .9275, Y 1.709

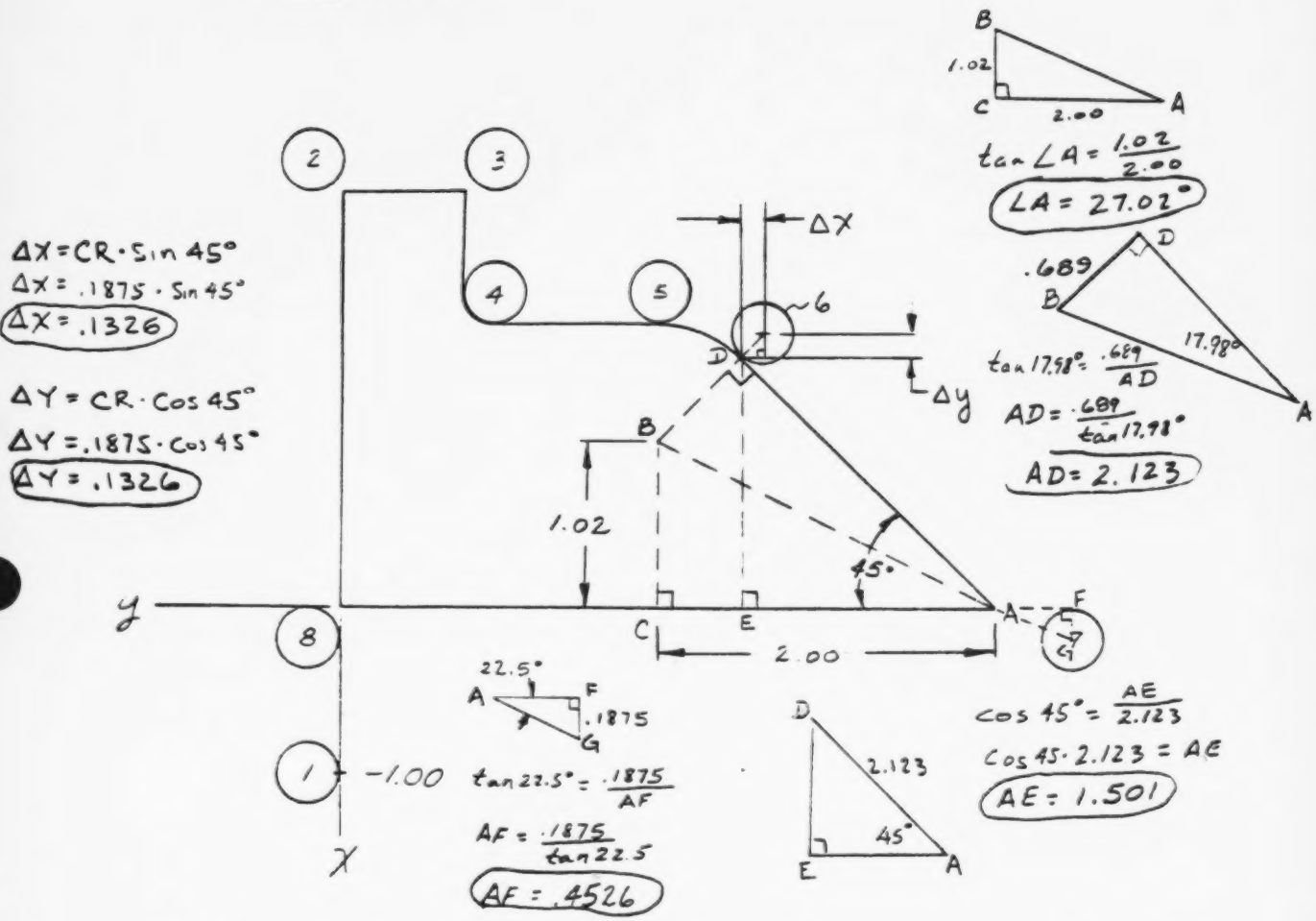
5) X 1.900, Y 1.709

6) X 2.5316, Y 1.6336

7) X 4.3527, Y -.1875

8) X -.1875, Y -.1875

b. Work solutions



5. Demonstrate procedure for tool path manual drawing.

B. Suggested student activities

1. Solve trigonometry problems, showing all work.

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)

UNIT III: NC MATH

LESSON 1: BASIC TRIGONOMETRY FOR NC

I. Cutter path construction **(TM 1.1, 1.2)**

A. The use of trigonometry is necessary. A thorough understanding of the use of right-angle concepts is a must.

B. The programmer is called upon to construct the cutter path. The NC machine operator must be able to distinguish the cutter path from the engineering blueprint.

1. This construction requires an awareness of the relationship of the workpiece's periphery to the center of the cutter.

2. The distance from the center of the cutter to the edge of the workpiece is the cutting tool **radius**--the core concept of center-line data constructed by the programmer.

C. Center-line data, which is the blueprint data converted into an NC sketch, is created to show the planned path of the cutter.

1. The programming of center-line data for tool positioning can be accomplished by two methods: absolute and incremental.

2. Both the absolute and the incremental methods utilize the Cartesian coordinate system.

3. C-L is an abbreviation for center-line data.

II. Right-angle trigonometry concepts **(HO 1.1-1.3)**

A. To solve a trigonometry problem, one must have a minimal knowledge about the triangle in question.

1. Minimum information is the length of two sides of the triangle or the length of one side plus one angle.

2. The sine of an angle is equal to the side opposite the angle divided by the hypotenuse of the triangle.

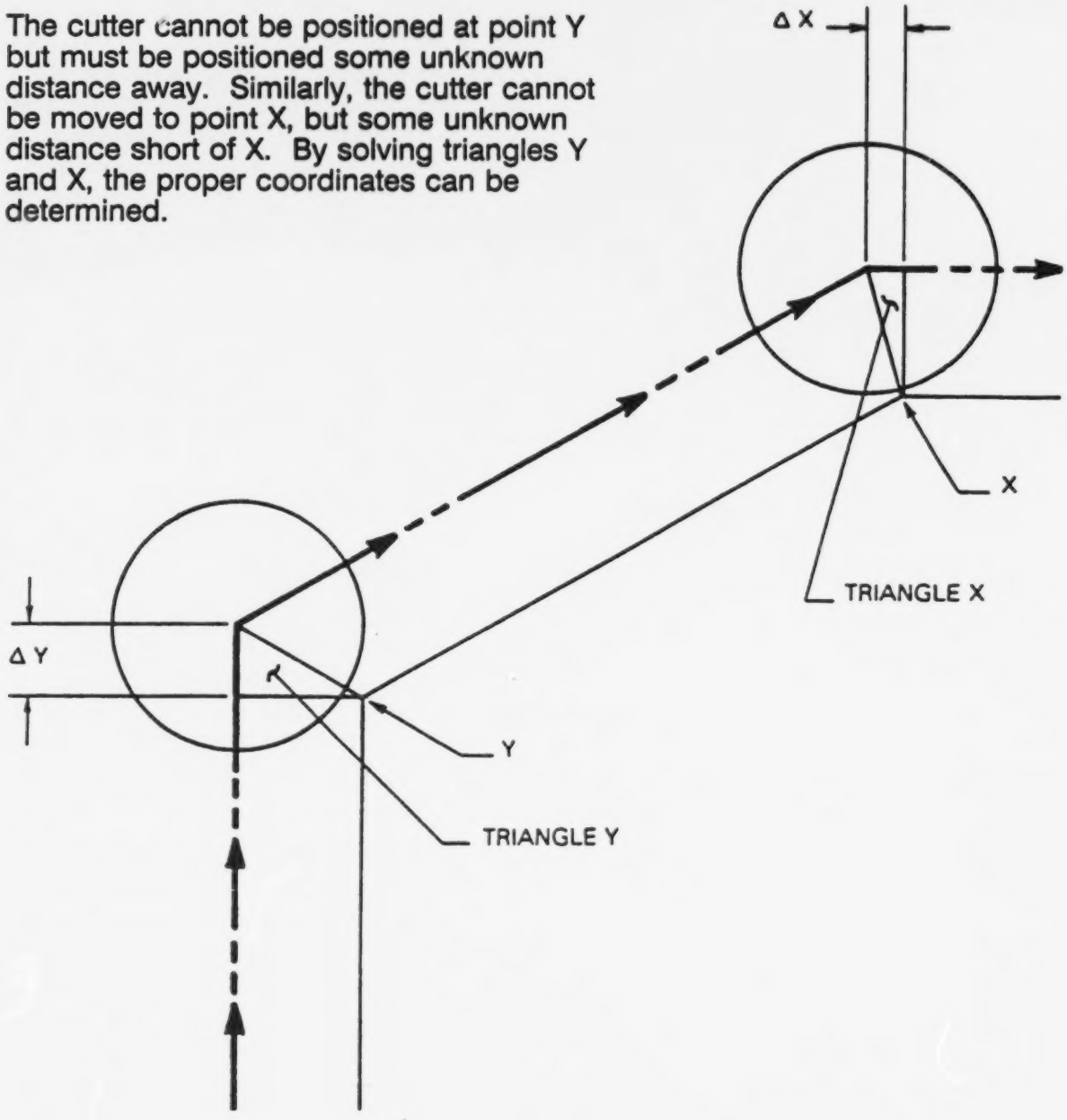
3. The cosine of an angle is equal to the side adjacent the angle divided by the hypotenuse of the triangle.

4. The tangent of an angle is equal to the side opposite the angle divided by the side adjacent the angle.

B. Situations do not always provide a simple right-angle trigonometry construction. When this is the case, oblique-angled triangulation is useful.

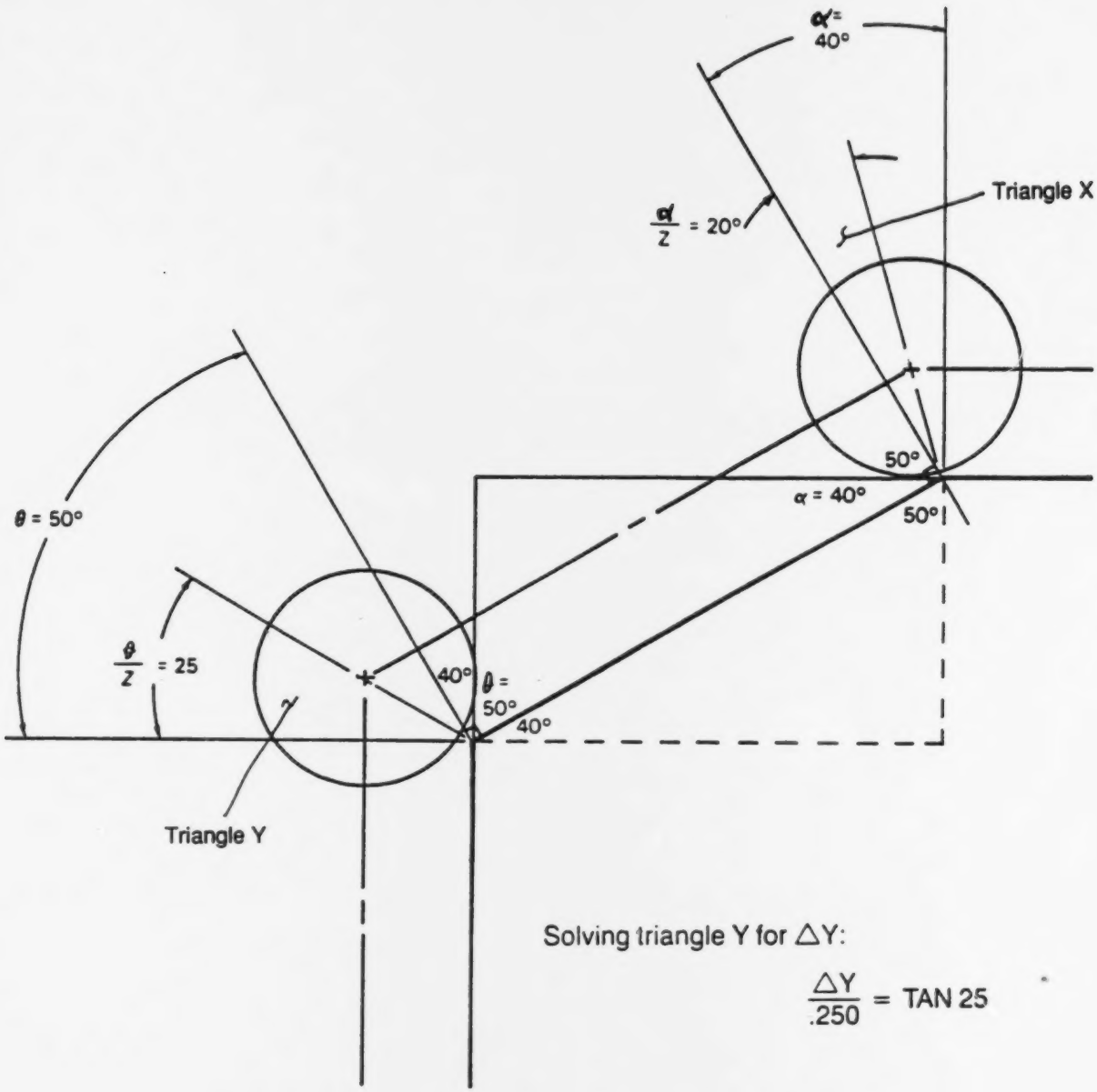
UNDERSTANDING CUTTER CENTER-LINE OFFSETS

The cutter cannot be positioned at point Y but must be positioned some unknown distance away. Similarly, the cutter cannot be moved to point X, but some unknown distance short of X. By solving triangles Y and X, the proper coordinates can be determined.



Credit: Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar, © 1986, pg. 164. Reproduced by permission.

UNDERSTANDING CUTTER CENTER-LINE OFFSETS (CONT.)

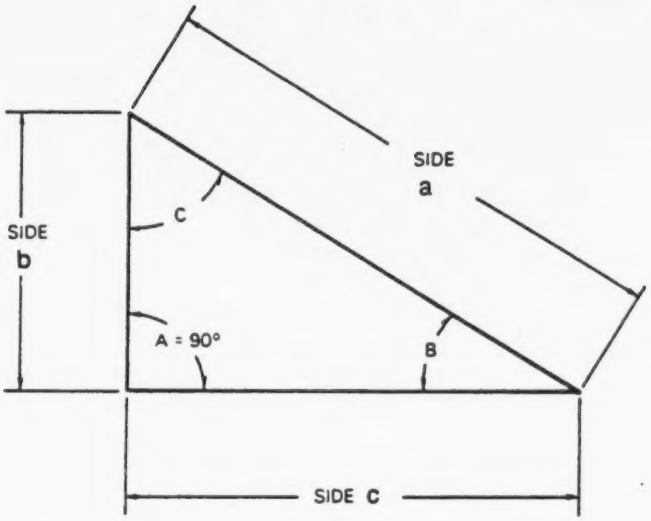


Solving triangle Y for △Y:

$$\frac{\triangle Y}{.250} = \text{TAN } 25$$

Credit: Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar, © 1986, pg. 165. Reproduced by permission.

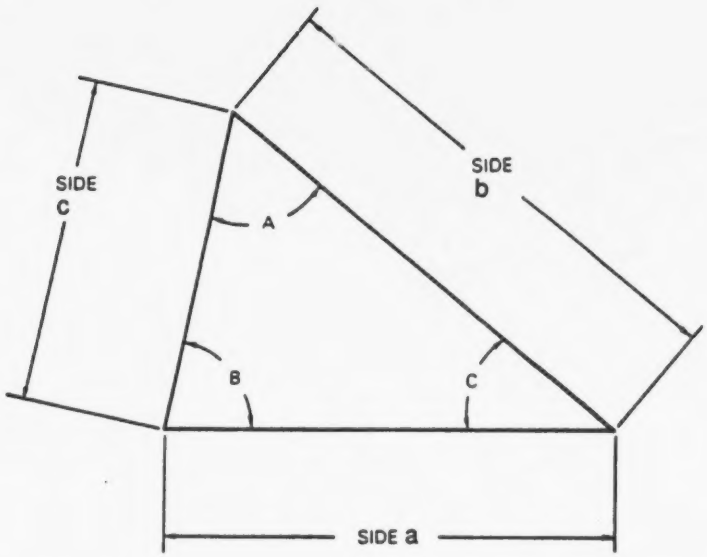
RIGHT TRIANGLES



KNOWN VARIABLES	SOLUTION FORMULAS		
SIDE a, ANGLE B	$b = a \times \sin B$	$c = a \times \cos B$	$C = 90° - B$
SIDE a, ANGLE C	$b = a \times \cos C$	$c = a \times \sin C$	$B = 90° - C$
SIDE b, ANGLE B	$a = \frac{b}{\sin B}$	$c = b \times \cot B$	$C = 90° - B$
SIDE b, ANGLE C	$a = \frac{b}{\cos C}$	$c = b \times \tan C$	$B = 90° - C$
SIDE c, ANGLE B	$a = \frac{c}{\cos B}$	$b = c \times \tan B$	$C = 90° - B$
SIDE c, ANGLE C	$a = \frac{c}{\sin C}$	$b = c \times \cot C$	$B = 90° - C$
SIDES a AND b	$c = \sqrt{a^2 - b^2}$	$\sin B = \frac{b}{a}$	$C = 90° - B$
SIDES a AND c	$b = \sqrt{a^2 - c^2}$	$\sin C = \frac{c}{a}$	$B = 90° - C$
SIDES b AND c	$a = \sqrt{b^2 + c^2}$	$\tan B = \frac{b}{c}$	$C = 90° - B$

Credit: Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar, © 1986, pg. 167. Reproduced by permission.

OBLIQUE TRIANGLES



ONE SIDE AND TWO ANGLES KNOWN:
GIVEN: SIDE a, OPPOSITE ANGLE A, AND OTHER ANGLE B

$$C = 180° - (A + B) \qquad b = \frac{a \times \text{SIN } B}{\text{SIN } A} \qquad c = \frac{a \times \text{SIN } C}{\text{SIN } A}$$

TWO SIDES AND THE ANGLE BETWEEN THEM KNOWN:
GIVEN: SIDES a, b, AND ANGLE C

$$\text{TAN } A = \frac{a \times \text{SIN } C}{b - (a \times \text{COS } C)} \qquad B = 180° - (A + C) \qquad c = \frac{a \times \text{SIN } C}{\text{SIN } A}$$

$$c = \sqrt{a^2 + b^2 - (2ab \times \text{COS } C)}$$

TWO SIDES AND ANGLE OPPOSITE ONE SIDE KNOWN:
GIVEN: SIDE a, OPPOSITE ANGLE A, AND SIDE B

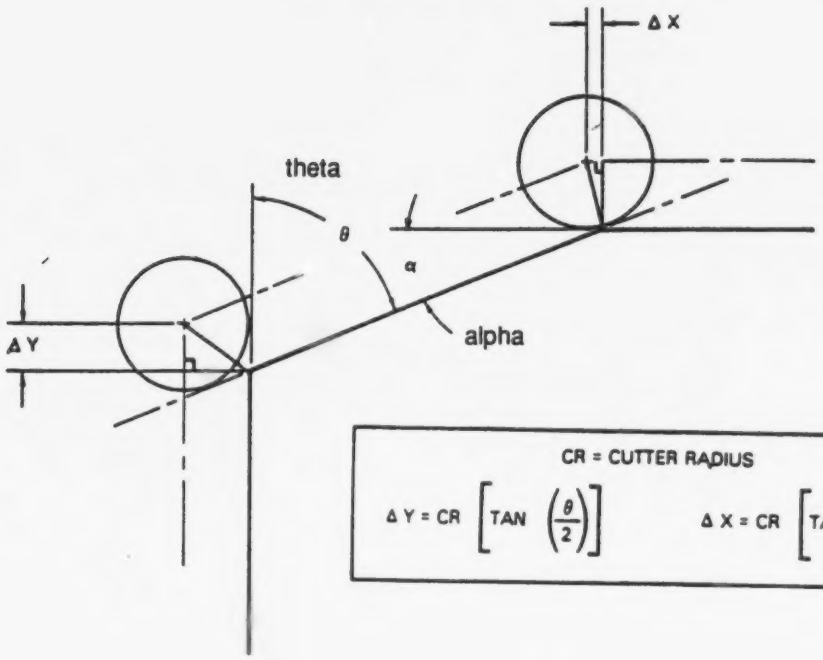
$$\text{SIN } B = \frac{b \times \text{SIN } A}{a} \qquad C = 180° - (A + B) \qquad c = \frac{a \times \text{SIN } C}{\text{SIN } A}$$

ALL THREE SIDES KNOWN:

$$\text{COS } A = \frac{b^2 + c^2 - a^2}{2bc} \qquad \text{SIN } B = \frac{b \times \text{SIN } A}{a} \qquad C = 180° - (A + B)$$

Credit: Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar, © 1986, pg. 168. Reproduced by permission.

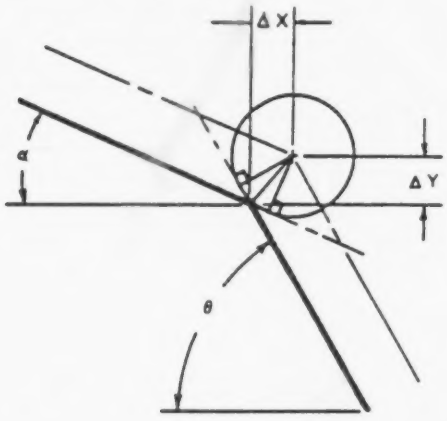
CENTER-LINE OFFSETS

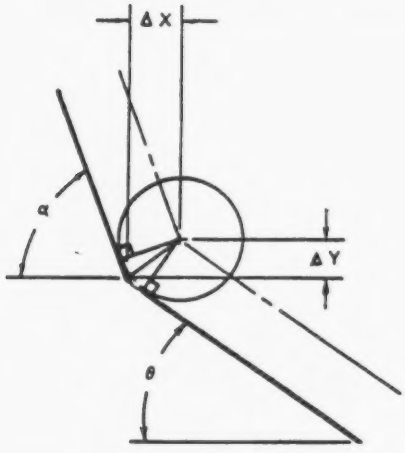


CR = CUTTER RADIUS

$$\Delta Y = CR \left[TAN \left(\frac{\theta}{2} \right) \right] \qquad \Delta X = CR \left[TAN \left(\frac{\alpha}{2} \right) \right]$$

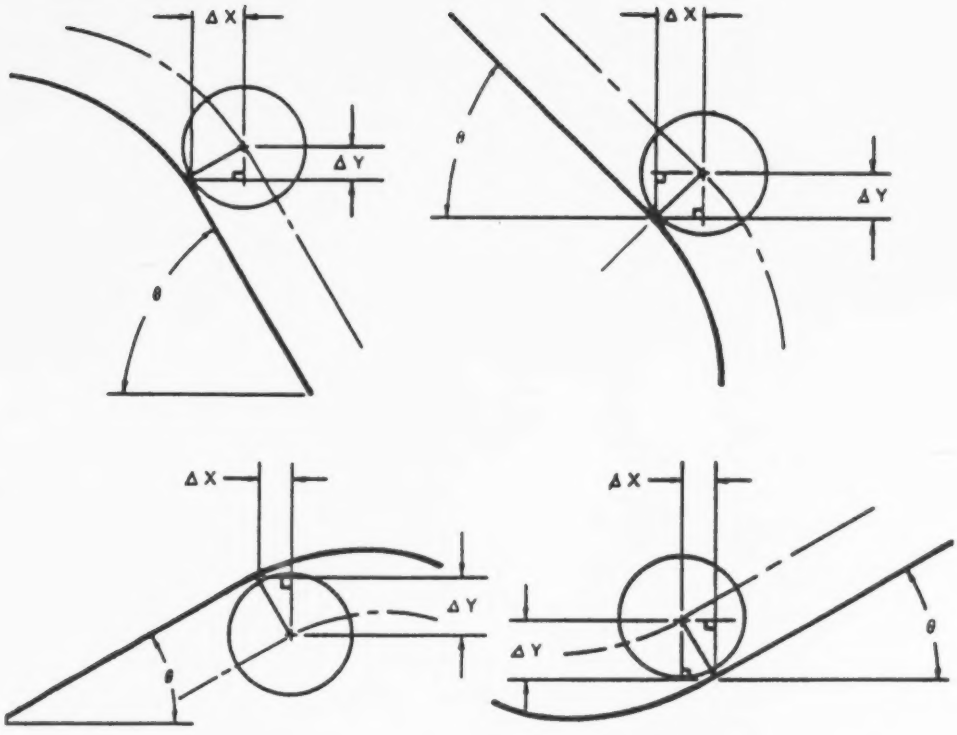
Two lines intersecting, not parallel to a machine axis





CR = CUTTER RADIUS

$$\Delta X = CR \times \frac{SIN\left(\frac{\alpha + \theta}{2} \right)}{COS\left(\frac{\alpha - \theta}{2} \right)} \qquad \Delta Y = CR \times \frac{COS\left(\frac{\alpha + \theta}{2} \right)}{COS\left(\frac{\alpha - \theta}{2} \right)}$$



CR = CUTTER RADIUS	
Δ X = CR X SIN θ	Δ Y = CR X COS θ

Line not parallel to a machine axis tangent to a circle

Credit: Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar, © 1986, pg. 169 and 170. Reproduced by permission.

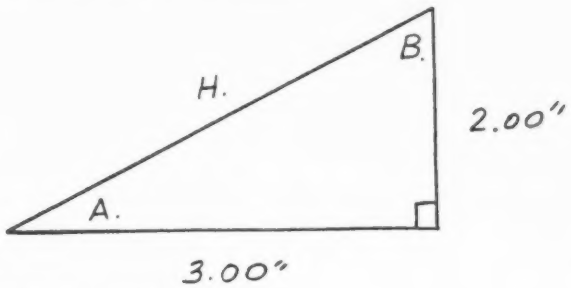
SIMPLE TRIGONOMETRY

Match the terms on the right with the appropriate definitions and mathematical solutions.

___ 1. The tangent of an angle is equal to ___ .

___ 2. The cosine of an angle is equal to ___ .

___ 3. The sine of an angle is equal to ___ .



___ 4. Express the formula to find H.

___ 5. Solve for H.

___ 6. Express the formula to find the tangent of A.

___ 7. Solve for angle A.

___ 8. Solve for angle B.

___ 9. The angle opposite H

___ 10. The length of the leg adjacent angle A

a. 33.69 deg.

b. opp. ÷ adj.

c. 3.00 in.

d. opp. ÷ hyp.

e. $a^2 + b^2 = c^2$

ab. adj. ÷ hyp.

ac. 3.606

ad. 56.31 deg.

ae. 90 deg.

bc. 6.00

ANSWERS TO ASSIGNMENT SHEET 1.1

1. b
2. ab
3. d
4. e
5. ac
6. b
7. a
8. ad
9. ae
10. c

MODULE: CNC (COMPUTERIZED NUMERICAL CONTROL)

UNIT III: NC MATH

UNIT TEST

Read the following carefully, then mark "a" for true or "b" for false.

____ 1. The distance from the center-line of the cutter to the periphery of the workpiece is the tool diameter.

____ 2. The programmer constructs the cutter path.

____ 3. Knowledge of trigonometry is not necessary to understand C-L data.

____ 4. Center-line data and cutter path data are usually the same.

____ 5. The NC sketch describes the C-L data.

____ 6. Absolute positioning is more accurate than incremental positioning.

____ 7. Blueprint data is more precise than the NC sketch.

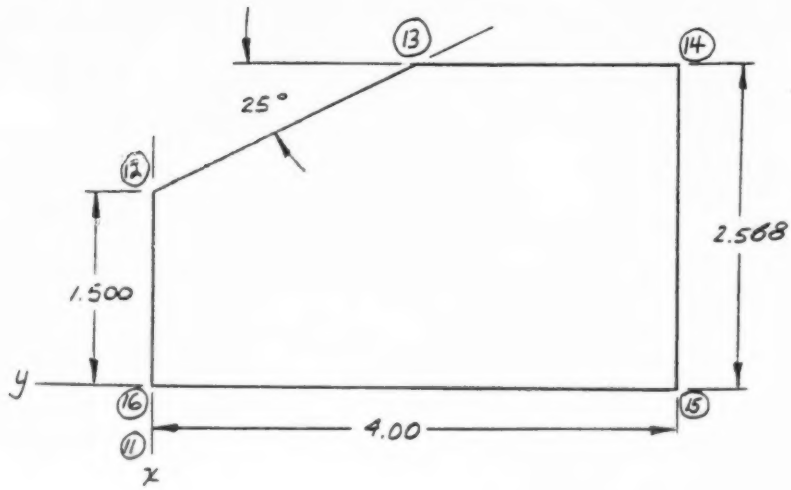
____ 8. The radius of the cutter is only as important as the machine operator's level of experience.

____ 9. The machine operator constructs the true C-L data.

____ 10. The planned path of the cutter is the NC sketch.

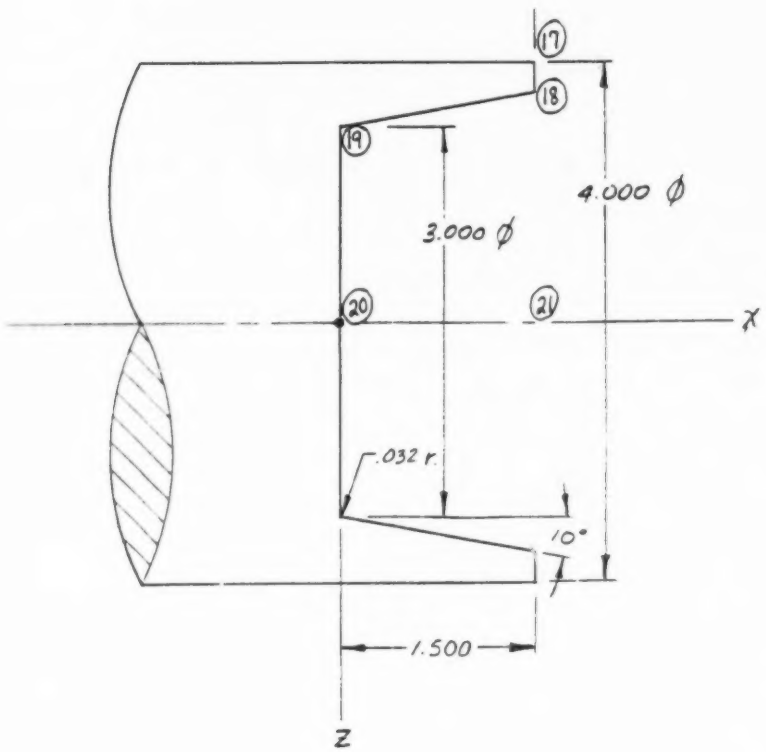
Using a 1/2" diameter end mill, determine the coordinate position for following center-line offset. Match the answer to the appropriate number. Show all work.

____	11.	a.	X -.250, Y -.250	ab.	X 1.6546, Y 2.818
____	12.	b.	X -.250, Y -1.00	ac.	X 3.250, Y 2.818
____	13.	c.	X -.250, Y 1.659	ad.	X 4.250, Y 2.818
____	14.	d.	X 1.100, Y 1.958	ae.	X 4.250, Y -.250
____	15.	e.	X 1.550, Y 2.525		
____	16.				



The five coordinate positions detailed below represent an imaginary diameter of 1/16" at the tip end of a lathe cutting tool. After calculating coordinates below, match answers to the appropriate number. Show all work.

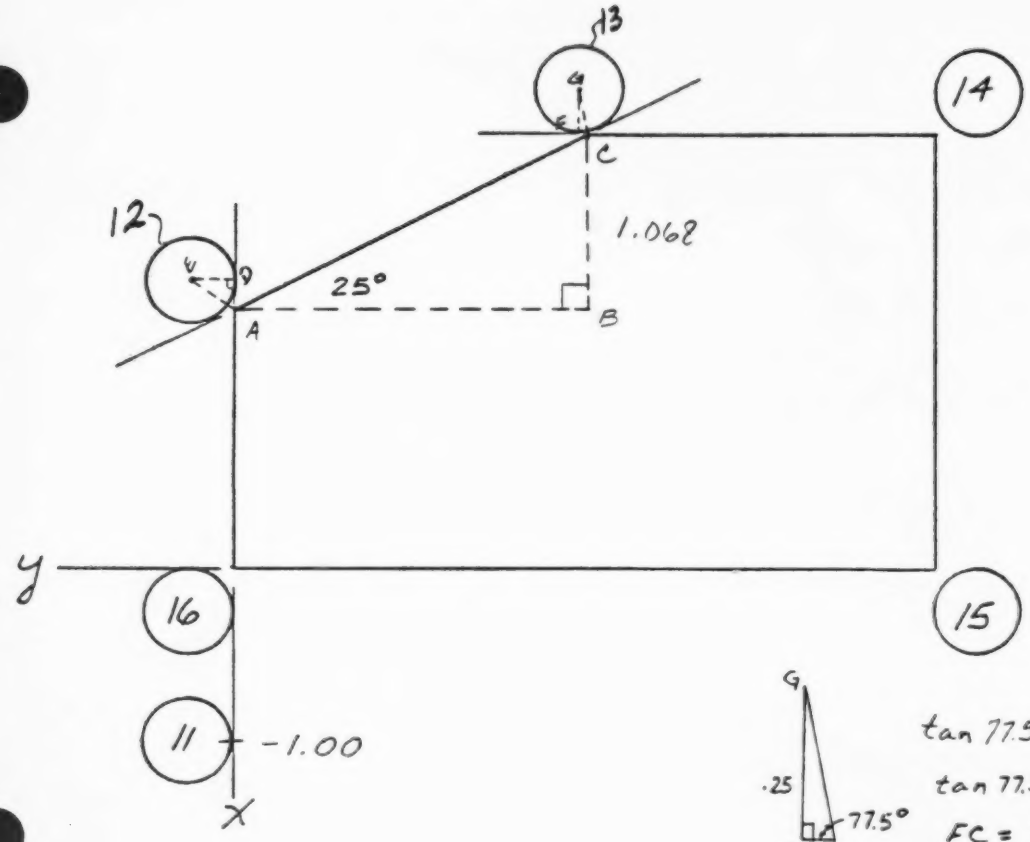
____	17.	a.	X 0, Z .023	ab.	X 1.6375, Z 1.532
____	18.	b.	X 0, Z .032	ac.	X 1.8375, Z 1.532
____	19.	c.	X 0, Z .0932	ad.	X 2.100, Z 1.532
____	20.	d.	X 0, Z 1.532	ae.	X 2.100, Z 1.932
____	21.	e.	X 1.4731, Z .032		

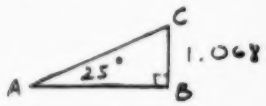


ANSWERS TO UNIT TEST

1.	b	(M4)
2.	a	
3.	b	(M4)
4.	a	(M4)
5.	a	(M4)
6.	b	(M4)
7.	b	
8.	b	
9.	b	(M4)
10.	a	(M4)
11.	b	(M4)
12.	c	(M4)
13.	ab	(M4)
14.	ad	(M4)
15.	ae	(M4)
16.	a	(M4)
17.	ad	(M4)
18.	ab	(M4)
19.	e	(M4)
20.	b	(M4)
21.	d	(M4)

INSTRUCTOR'S KEY



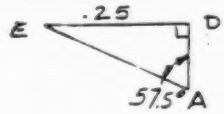


$\tan 25° = \frac{1.068}{AB}$

$\tan 25° \cdot AB = 1.068$

$AB = \frac{1.068}{\tan 25°}$

$AB = 2.290$

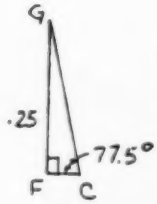


$\tan 57.5° = \frac{.25}{AD}$

$\tan 57.5° \cdot AD = .25$

$AD = \frac{.25}{\tan 57.5}$

$AD = .159$

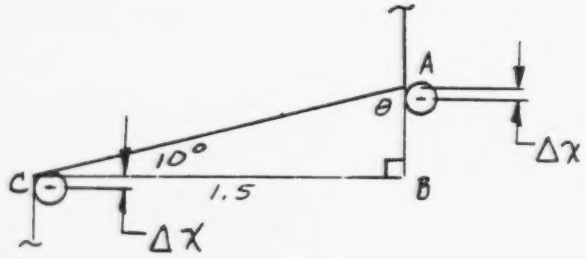


$\tan 77.5° = \frac{.25}{FC}$

$\tan 77.5° \cdot FC = .25$

$FC = \frac{.25}{\tan 77.5°}$

$FC = .0554$



$\Delta X = CR\left[\tan\left(\frac{\theta}{2}\right)\right]$

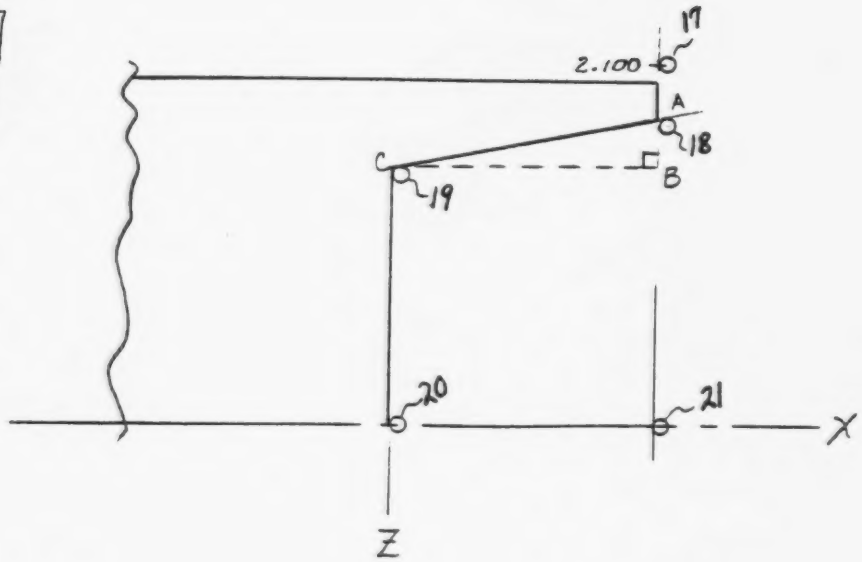
$\Delta X = .032\left[\tan\left(\frac{80°}{2}\right)\right]$

$\Delta X = .0269$

$\tan 10° = \frac{AB}{1.5}$

$AB = \tan 10° \cdot 1.5$

$AB = .264$



MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL
UNIT IV: PROGRAMMING FOR NC MACHINES

UNIT OBJECTIVE

Upon completion of this unit, the student should be able to understand detail the processes and create a shop book of the NC operations. Understand and identify G codes and M functions. This knowledge will be evidenced by achieving the specified score on assignment sheets and the unit test.

SPECIFIC OBJECTIVES

After completion of the lessons in this unit, the student should be able to:

1.1 Distinguish between methods of workpiece dimensioning.

1.2 Solve problems concerning speed and feed rates.

2.1 Identify aspects of the manufacturing plan.

2.2 Identify components of a tool inventory sheet.

2.3 Describe operation set-up sheet purposes.

2.4 Describe operation instruction sheet uses.

3.1 Match preparatory functions (G codes) to uses.

3.2 Match miscellaneous functions (M codes) to uses.

4.1 Identify the functions of the programmer. (competency M4)

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT IV: PROGRAMMING FOR NC MACHINES

CONTENTS OF THIS UNIT

A. Objective sheet

B. References

C. Notes to the instructor

D. Lesson plans

1. Lesson 1: VISUALIZE MACHINING SEQUENCES

   a. Information outline

   b. Assignment sheets

      AS 1.1: Workpiece problems
      AS 1.2: Tooling, speed and feed

   c. Answers to assignment sheets

2. Lesson 2: THE SHOP BOOK

   a. Information outline

   b. Handouts

      HO 2.1: The manufacturing process
      HO 2.2: Tool inventory
      HO 2.3: Operation set-up
      HO 2.4: Operation instructions

   c. Assignment sheet

      AS 2.1: Shop book problems

   d. Answers to assignment sheet

3. Lesson 3: FUNCTION CODES

   a. Information outline

   b. Assignment sheet

      AS 3.1: Function code problems

   c. Answers to assignment sheet

4. Lesson 4: FUNCTIONS OF THE PROGRAMMER

   a. Information outline

E. Unit test

F. Answers to unit test

REFERENCES

A. Books

1. Luggen, William W. Fundamentals of Numerical Control. 2nd ed. Albany, NY: Delmar Publishers, Inc., 1988.

2. Roberts, Arthur D. Programming for Numerical Control Machines. 2nd ed. New York: McGraw Hill, 1978.

3. Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar Publishers, Inc., 1986.

4. Stranton, George C. Numerical Control Programming. 1st ed. New York: John Wiley & Sons, Inc., 1988.

B. Others

1. Computerized Numerical Control (test item bank). Atlanta, GA: Vocational-Technical Education Consortium of States; Carbondale, IL: Dept. of Vocational Education Studies, Southern Illinois University, 1986.

2. Theories and Operation of Numerical Control/Computerized Numerical Control (NC/CNC) (U105001). Urbana: University of Illinois at Urbana-Champaign: Vocational Education Service, March 1988.

NOTES TO THE INSTRUCTOR

A. Suggested approaches to teaching this unit

1. Acquire information from manufacturers that would be helpful to students.

2. Demonstrate preparation of operator instructions.

3. Demonstrate how to write programmed instruction.

4. Discuss the time/cost factors for different types of programming.

5. Handouts 2.1-2.4 illustrate plans for machining pen holders. The author has used the initials "LTC" (Linn Technical College); alter lettering and numbering to be appropriate for your school or allow students to design their own.

B. Suggested student activities

1. Given a workpiece blueprint, the student will prepare a part program and related documentation for the operator.

2. Allow students to review shop book handouts until time for the unit test.

C. Suggested supplemental materials

1. Oberg, Erik; Franklin D. Jones; Holbrook L. Horton. Machinery's Handbook. 22nd ed. New York: Industrial Press Inc., 1984.

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT IV: PROGRAMMING FOR NC MACHINES

LESSON 1: VISUALIZE MACHINING SEQUENCES

I. Determining workpiece dimensions

A. The first step in programming is to visualize the machining processes while reviewing the blueprint.

B. Front, top and side views are then developed, based on processes.

C. All dimensions should be from one common datum.

D. The sketch must provide programmer with dimension information for developing programmed instructions for the workpiece's production.

E. Determine degree of shape complexity for the program.

1. Simple shape--use manual program

2. Complex shape--use computer

F. Adapt machining process to combine operations where possible.

G. Match machining requirements to minimize labor and machine tool operating costs.

H. Establish reference point to align workpiece and machine tool.

I. Determine stock material to be machined from.

J. Specify holding device location in reference to table and workpiece.

K. Select incremental mode if accuracy and tolerance do not present any problems or if little or no contour milling is required.

L. With the following situations, select the absolute mode, the most widely used and accepted of the two dimensioning methods.

1. Close tolerances and high accuracy required

2. Complex contour milling

3. When determining cutting tool location in relation to specific datum point

II. Tool selection, speeds and feeds

A. Cutting tools

1. Cutting tool selected must create workpiece as specified on print. Tool holder must fit cutting tool.

2. Check cutting tool dimensional standards for exact tolerance.

3. Select the shortest and most rigid cutting tool that can be applied to job.

4. Utilize commercial, pre-set or qualified types of tools on lathe operations.

5. Identify tools and workpiece holders on the basis of type, specific cutting edge required for material, diameter and maximum tool life.

6. Establish machine tool loading and unloading requirements for workpiece.

7. If available, select automatic tool changers for multiple cutting tool operations.

B. Speeds and feeds

1. Determine an acceptable recommended spindle speed by calculation and table reference.

2. Identify material cutting rate from the Machinery's Handbook.

3. Convert cutting rates to revolutions per minute.

   [formula: rpm (revolutions per minute) = sfm (surface feet per minute) x constant (3.1416) ÷ D (diameter of cutter).]

4. Calculate axis travel feed rate in inches per minute or millimeters for "spindle plunge" and contour milling operations.

5. Convert feed to feed rate using the following formulas.

   a. Drilling processes: Feed rate = feed x spindle speed; inches per minute = inches per revolution x revolutions per minute (ipm = ipr x rpm). If using metrics, substitute the metric dimension.

   b. Milling process: Feed in inches per minute is equal to feed per tooth in inches times number of teeth per cutter times revolutions per minute (F = f x T x rpm).

6. Modify standard cutting speed when necessary.

   a. Lower speeds for heavy cuts, maximum cutter life, minimum tool wear or materials that are hard, tough or abrasive.

   b. Use higher speeds for light cuts, softer or non-metallic materials, frail workpieces or set-up, maximum production rate, better finishes and high-helix cutters.

WORKPIECE PROBLEMS

Write "a" before true statements and "b" before false ones.

____ 1. All dimensions should be from one common datum.

____ 2. Machining processes should be combined where possible.

____ 3. The top view is often unnecessary and should be omitted.

____ 4. The type of material to be machined is not important at the "visualization" stage of planning.

____ 5. When accuracy and tolerance are unimportant, the incremental milling mode is the best to use.

____ 6. The absolute mode determines the cutting tool location from a specific datum point (origin).

____ 7. The incremental milling mode is the most widely used of the two modes.

____ 8. Matching the machine's capacity and the project profit is a must.

____ 9. Contouring work is best done using the incremental mode of milling.

____ 10. Holding devices must always be placed only at locations specified by the programmer.

TOOLING, SPEED AND FEED

Select the option which best answers the question or completes the statement.

___ 1. For greater efficiency the cutting tool should be _____ .

a. A right-hand fast spiral
b. One-half the slot size
c. Rigid and short as possible
d. A left-hand slow spiral

___ 2. Lathe tooling is more economical if a _____ type is used.

a. Custom (for specific job)
b. High-speed
c. Special order
d. Commercial, pre-set

___ 3. Automatic tool changers are useful when _____ .

a. Preset tooling is used
b. Multiple tool operations are involved
c. No set length is required
d. Pressed for time

___ 4. Before programming a precision tolerance cut on the workpiece, the programmer should check _____ .

a. With the operator
b. With the boss
c. Quality control
d. Cutting tool dimensional standards

___ 5. When determining the appropriate spindle speed, the programmer _____ .

a. Converts cutting rates to rpms
b. Divides cutting speed by surface feet per minute
c. Uses great imagination
d. Uses the same cutting speed for all work

___ 6. The higher the spindle speed of the machine, the _____ .

a. Better hard material can be cut
b. Tougher the stock should be
c. Faster the production rate
d. Longer a cutter will last

___ 7. When modifying standard cutting speeds, the programmer can achieve better results on soft materials by _____ .

a. Using higher speeds
b. Using coolant
c. Using slower speeds
d. a & b
e. None of the above

___ 8. High-helix cutters are best suited for _____ .

a. Minimum tool wear
b. Frail workpiece or set-up
c. Better finishes
d. a & c
e. b & c

___ 9. The feed rate for a drilling operation can be determined by _____ .

a. Adding the feed to the cutting speed
b. Dividing feed rate by spindle speed
c. Multiplying the spindle speed by the feed
d. Subtracting rpms from the ipr

___ 10. The length of time needed to mill a workpiece can be calculated by _____ .

a. Multiplying the stock removal by the speed the stock is removed
b. Adding the length of cut to the feed rate
c. Multiplying the feed rate by the spindle speed
d. Converting feed rate into inches

ANSWERS TO AS 1.1

1. a
2. a
3. b
4. b
5. a
6. a
7. b
8. b
9. b
10. b

ANSWERS TO AS 1.2

1. c
2. d
3. b
4. d
5. a
6. b
7. d
8. e
9. c
10. a

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT IV: PROGRAMMING FOR NC MACHINES

LESSON 2: THE SHOP BOOK

NOTE: A well-kept shop book, detailing each step of the machining process, is especially important in a large business when more than one shift works on the same workpiece. Clear, written plans provide continuity and prevent misunderstandings and misinterpretations. A small, one-person shop, however, may not find it as useful to keep records in such detail. When repeatability of performance is desired, an accurate record of each process is a must. The shop book also serves as a reliable method for quality control.

I. Manufacturing plan **(HO 2.1)**

A. A set of drawings are created to describe the machining process.

B. These drawings illustrate stock being removed at various stages.

C. Each sequence is color-coded and matched with the cutter required for stock removal.

D. Every tooling change separates another sequence of material removal.

E. These sequences are referred to as pages and each page number is called out in the programmer's manuscript.

F. Also identified on each manufacturing plan is the machine on which the workpiece is machined, the planner and programmer, and the date.

II. Tool inventory sheet **(HO 2.2)**

A. The list of cutting tools must list the type and quantity of cutting tools needed.

B. Each tool corresponds with the tool cutter description in the manufacturing plan.

C. T.No. designates which position the tool is placed in the automatic tool changer.

D. Tool offsets can be used when compensating for undersized cutters. When full size cutters are used, record the word "none".

E. Tool holders should be listed by manufacturing specifications.

F. When preset tooling is to be used, the planner/programmer must indicate the necessary set length.

G. The tool inventory sheet identifies the part number and name, the planner/programmer and the date.

III. Operation set-up sheet **(HO 2.3)**

NOTE: In some machine shops the operator has full responsibility for locating the workpiece on work table. If so, it is impractical for the programmer to speculate where the workpiece will be held down. This sheet can then be omitted.

A. The programmer must specify how the workpiece is to be held to the table and its position from machine "home."

B. Set-up sheets are necessary when any new clamping or fixture is required. If only one work-holding arrangement is necessary, only one set-up sheet is used.

C. The distance from home or machine zero to part zero must be referenced.

D. The accumulated height of the fixture or parallels plus the workpiece thickness is indicated in a front view.

E. Distances programmed to reference point returns and gage heights from the workpiece are described on the set-up sheet front view.

IV. Operation instruction sheet **(HO 2.4)**

A. Each tool change is accompanied by an operation instruction sheet.

B. The cutting tool diameter, number of flutes and set length should be identical to that called out on the tool inventory sheet.

C. Occasionally the programmer is called upon to program both a left- and right-hand part. Time is saved by using the invert switch on some machines, thereby using the same programmed data.

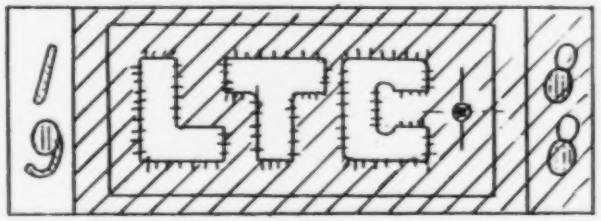
D. The operator must be notified of the direction of spindle rotation (clockwise or counter-clockwise).

E. Other significant data must be communicated to the machine operator, such as the programmed rpm and the feed rate range.

F. The procedure noted on each operation instruction sheet can be followed years later. Using the same data and set-up, the expected results will be repeated and guesswork eliminated.

MACHINE : KIRA 3 AXIS PLANNER DHC PROGRAMMER DHC

PAGE NO.	CUTTER DESCRIPTION	DESCRIPTION OF OPERATION
08	2.00 x .5 x .12 x 2	ROUGH MILL & SPOT FACE
09	.5 x .25 x .06 x 2	FINISH MILL AROUND LETTERS
12	29/64 TAP DRILL	DRILL HOLE
13	1/4 - 20 TAP	TAP HOLE
14	2.00 x .687 x .12 x 2	SPOT FACE NUMBERS
15	.375 x .125 x .06 x 2	MILL NUMBERS



PAGE 1 OF 1

10/2/87 DHC

HO 2.2

CNC 121

TOOL INVENTORY

PART NO. 546

PART NAME PEN HOLDER BASE PLANNER DHC. PROGRAMMER DHC

T. NO.	QTY	TOOL DESCRIPTION	TOOL OFFSET	TOOL HOLDER	SET LENGTH
08	1	2.00 FL × .5 DIA × .12 CR × 2 FLT	NONE	STD. EXTENSION	NONE
09	1	.5 FL × .25 DIA × .06 CR × 2 FLT	NONE	STD. EXTENSION	NONE
12	1	29/64 TAP DRILL	NONE	STD. COLLET	NONE
13	1	1/4 TAP	NONE	STD. TAPPING HD.	NONE
14	1	2.00 FL × .687 DIA × .12 CR × 2 FLT	NONE	STD.	NONE
15	1	.375 FL × .125 DIA × .06 CR × 2 FLT	NONE	STD. EXTENSION	NONE

PAGE 1 OF 1

10/24/87 D.H.C.

HO 2.3

CNC 123

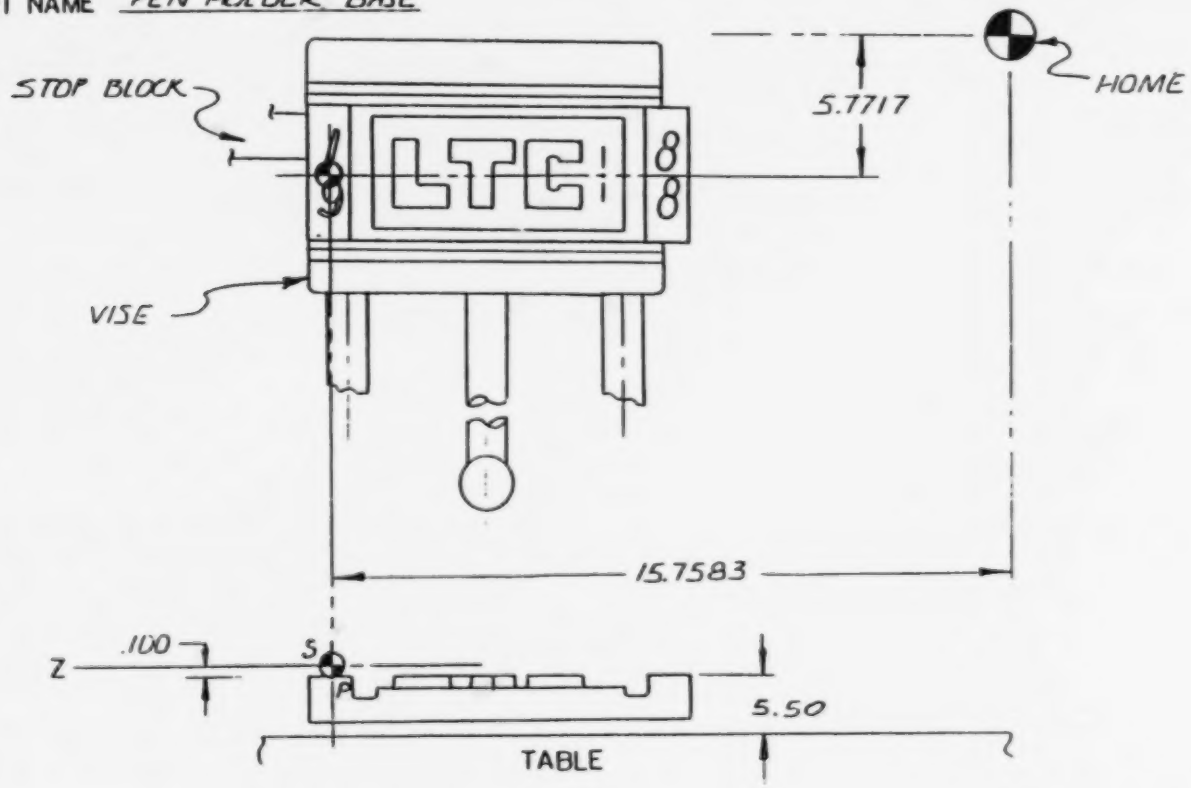
MATERIAL AL. DIE CAST ALLOY
PART NO. 546
PART NAME PEN HOLDER BASE

PLANNER DHC

PROGRAMMER DHC

DATE 10/24/87

OPERATION SET-UP



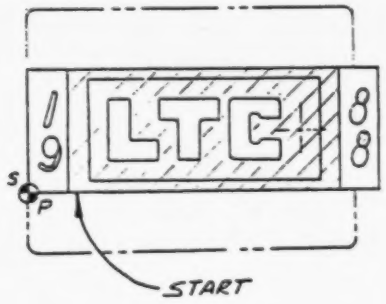
PAGE 1 OF 1

10/24/87 D.H.C.

OPERATION INSTRUCTIONS

PART NO 546
PART NAME PEN HOLDER BASE CUTTER DIA. .50 FLUTES 2 SET LENGTH NONE
INVERT SWITCHES ~ X- N Y- N Z - N SPINDLE ROTATION CW RPM 800
FEED RATE RANGE F8.0

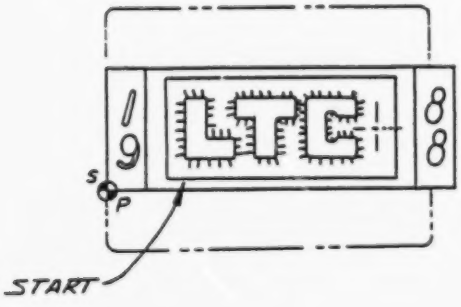
PROCEDURE LOAD CUTTER~ ZERO RETURN~ LOCATE Z DEPTH TO PART SURF. ENTER TOOL OFFSET VALUE @ PARAMETER MENU ~ ZERO RETURN SET AUTOMATIC OPERATION~START ~ ROUGH CUT LETTERS~LEAVE .010 MAT'L. ~ SPOT FACE ~ STOP AT END OF PAGE. ~ CHANGE CUTTER



PAGE 1 OF 6

PART NO. 546
PART NAME PEN HOLDER BASE CUTTER DIA. .25 FLUTES 2 SET LENGTH NONE
INVERT SWITCHES ~ X- N Y- N Z - N SPINDLE ROTATION CW RPM 1600
FEED RATE RANGE F8.0

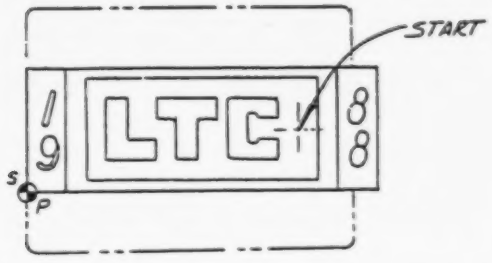
PROCEDURE LOAD CUTTER~ ZERO RETURN~ LOCATE Z DEPTH TO PART SURF. ENTER TOOL OFFSET VALUE @ PARAMETER MENU ~ ZERO RETURN SET AUTOMATIC OPERATION~START ~ FINISH MILL AROUND LETTERS ~ STOP AT END OF PAGE ~ CHANGE CUTTER



PAGE 2 OF 6

PART NO 546
PART NAME PEN HOLDER BASE CUTTER DIA. ____ FLUTES 2 SET LENGTH NONE
INVERT SWITCHES ~ X- N Y- N Z - N SPINDLE ROTATION CW RPM 3200
FEED RATE RANGE F25.0

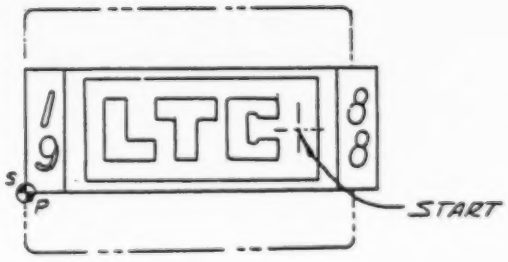
PROCEDURE LOAD CUTTER~ ZERO RETURN~ LOCATE Z DEPTH TO PART SURF. ENTER TOOL OFFSET VALUE @ PARAMETER MENU ~ ZERO RETURN SET AUTOMATIC OPERATION~START ~ DRILL HOLE ~ STOP AT END OF PAGE ~ CHANGE CUTTER



w/24/87 D.H.C.

PART NO 546
PART NAME PEN HOLDER BASE CUTTER DIA. ____ FLUTES 2 SET LENGTH NONE
INVERT SWITCHES ~ X- N Y- N Z - N SPINDLE ROTATION CW RPM 400
FEED RATE RANGE F200

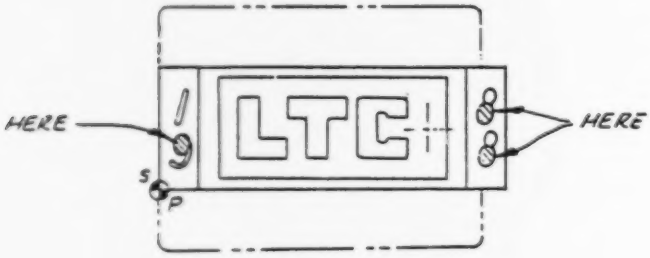
PROCEDURE LOAD CUTTER~ ZERO RETURN~ LOCATE Z DEPTH TO PART SURF. ENTER TOOL OFFSET VALUE @ PARAMETER MENU ~ ZERO RETURN SET AUTOMATIC OPERATION~START ~ TAP HOLE ~ STOP AT END OF PAGE ~CHANGE CUTTER



w/24/87 D.H.C.

PART NO 546
PART NAME PEN HOLDER BASE CUTTER DIA .687 FLUTES 2 SET LENGTH NONE
INVERT SWITCHES ~ X- N Y- N Z - N SPINDLE ROTATION CW RPM 1000
FEED RATE RANGE F15.0

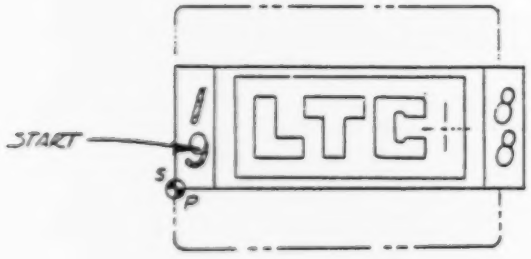
PROCEDURE LOAD CUTTER - ZERO RETURN - LOCATE Z DEPTH TO PART SURF. ENTER TOOL OFFSET VALUE @ PARAMETER MENU ~ ZERO RETURN SET AUTOMATIC OPERATION - START ~ SPOTFACE NUMBERS - STOP AT END OF PAGE - CHANGE CUTTER



11/24/87 D.H.C.

PART NO 546
PART NAME PEN HOLDER BASE CUTTER DIA ____ FLUTES 2 SET LENGTH NONE
INVERT SWITCHES ~ X- N Y- N Z - N SPINDLE ROTATION CW RPM 3200
FEED RATE RANGE F15.0

PROCEDURE LOAD CUTTER - ZERO RETURN - LOCATE Z DEPTH TO PART SURF. ENTER TOOL OFFSET VALUE @ PARAMETER MENU ~ ZERO RETURN SET AUTOMATIC OPERATION - START ~ MILL NUMBERS TO END OF BOOK.



11/24/87 D.H.C.

SHOP BOOK PROBLEMS

Write "a" before true statements and "b" before false ones.

____ 1. The number assigned to each cutting tool is its position on the automatic tool changer (ATC).

____ 2. Set length information is not needed with pre-set tooling.

____ 3. The area where stock is to be removed is illustrated on the manufacturing plan.

____ 4. Each tool change requires an operation instruction sheet.

____ 5. Tool offsets are used when cutters are undersized due to regrinding.

____ 6. Clamping and/or fixture locations are not important to the programmer.

____ 7. The operator should know which way the spindle rotates; therefore, the programmer need not specify it.

____ 8. Feed rate and spindle speed are recorded on the operation set-up sheet.

____ 9. Every program page has the same number as the program tool number.

ANSWERS TO AS 2.1

1. a
2. b
3. a
4. a
5. a
6. b
7. b
8. b
9. a

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT IV: PROGRAMMING FOR NC MACHINES

LESSON 3: FUNCTION CODES

I. Preparatory functions (G codes)

A. The preparatory function or cycle code is a two-digit number preceded by the word address letter G (G##).

B. The G code determines the mode of operation.

C. Various machine control units interpret different G codes. See manufacturer's manuals for specifics.

D. Linear interpolation is always identified as G01. This commands the slides (lathe) or table (mill) to move the tool in a straight line from the current position to the command position.

E. Circular interpolation is identified as G02 or G03. An arc can be machined by moving in a clockwise or counter-clockwise direction.

1. G02 indicates clockwise movement.

2. G03 indicates counter-clockwise movement.

F. Certain canned cycles (fixed or preset) can be accessed through G codes, such as G76 for boring, G81 for drilling and G84 for the tapping cycle.

II. Miscellaneous functions (M codes)

A. The word miscellaneous is somewhat misleading since these functions are vitally important to the machining process.

B. The miscellaneous function is a two-digit number preceded by the word address letter M (M##).

C. M codes are on-off commands for machine operations.

D. Spindle start, spindle stop, coolant on, coolant off, end of program (M30), program stop (M00) and jump to subroutine commands are accessed through the M codes.

FUNCTION CODE PROBLEMS

Match the terms on the right with the appropriate definitions.

____	1.	Command to start and stop the spindle	a. G code
____	2.	Preparatory function for linear interpolation	b. M code
____	3.	Code which determines the mode of operation	c. On-off commands
____	4.	To be reviewed when in doubt about interpreting preparatory functions	d. Canned cycle
____	5.	Another name for fixed cycle	e. G02
____	6.	Calls for the fixed cycle	ab. G01
____	7.	Command for the end of program and memory reset	ac. Machine builder's manual
____	8.	Command for clockwise circular interpolation	ad. M30
____	9.	Counter-clockwise circular interpolation	ae. G03
____	10.	Program stop	bc. M00

ANSWERS TO AS 3.1

1. b
2. ab
3. a
4. ac
5. d
6. a
7. ad
8. e
9. ae
10. bc

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT IV: PROGRAMMING FOR NC MACHINES

LESSON 4: FUNCTIONS OF THE PROGRAMMER

I. Programmer's responsibility

A. The NC programmer must possess many skills to get the job done.

B. There must be a strong knowledge of blueprint language and the ability to convert that language into an NC sketch.

1. The sketch provides the programmer with dimensioning information for developing programmed instructions for the workpiece's production.

2. Should a sample part be used in place of the blueprint, the programmer must have full command of all appropriate measuring tools to ascertain accurate dimensions.

C. A complete understanding of tolerances necessary for mating parts or precision fits is required.

D. The programmer must be able to develop an adequate method for holding the workpiece during machining operations.

E. The ability to discern which machining operation should come first is also important.

F. The most efficient schedule of operations must be visualized and implemented.

G. Safety should be always be a major concern.

H. Machine-down time means lost profits and high repair costs.

I. The programmer must be a logical and methodical thinker who tries to utilize machinery and staff in the best possible way.

J. The ability to read and understand charts concerning cutting tool specifications, materials, cutting speeds and feed rate recommendations, tool holders, material hardness and metric conversions is needed.

K. Basic geometry and trigonometry must be used efficiently.

L. A working knowledge of computer processes is desired.

M. The capability to manipulate computer data is indispensible.

N. The programmer must be fluent in the NC language or format used.

O. Prepared instructions must be developed to implement programmed machine commands.

P. Frequently, fixtures and clamps must be designed by the programmer.

Q. For each operation, the programmer selects preparatory and miscellaneous codes, offset register codes, axis coordinates, other specific machine codes and specific sequence for each line number.

R. After the initial programming, revisions may be needed due to engineering changes.

S. In addition, notification of all changes must be provided to significant departments.

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT IV: PROGRAMMING FOR NC MACHINES

UNIT TEST

Select the option which answers the question or completes the statement.

___ 1. Which formula is used when converting drilling feed to feed rate?

a. rpm = sfm x π ÷ D
b. ipm = ipr x rpm
c. F = f x T x rpm
d. T = L ÷ F
e. ipr = ipm x rpm

___ 2. What is a program manuscript?

a. General term which refers to any storage media for binary data
b. Portion of a computer exclusive of the input, output, peripherals and storage units
c. Logic matrix that describes a logic function by listing all possible combinations
d. Form used by a programmer for listing detailed manual or computer part programming instructions
e. Computer program stored in memory

___ 3. When converting cutting rate to revolutions per minute, the following formula is used (rpm = sfm x π ÷ D). What does the sfm stand for?

a. Surface feed per minute
b. Surface feet per meter
c. Surface feed per millimeter
d. Surface feet per minute
e. Standard feed per microsecond

___ 4. How is feed rate for axis travel usually expressed?

a. Centimeters or feet per minute
b. Meters or yards per minute
c. Millimeters or inches per minute
d. Decimeter or inches per minute
e. Binary-coded decimals

___ 5. Which type of interpolation which allows cutting tool to move in a path ranging from a small arc to a full 360-degree span?

a. Circular
b. Programmed
c. Lateral
d. Linear
e. None of the above

___ 6. Given a 200 sfpm and a 1" tool diameter, what is the rpm?

a. 191
b. 200
c. 382
d. 512
e. 764

___ 7. Given a feed per revolution of .006 and a 250 rpm, what is the feed per minute?

a. .75
b. 1.5
c. 6
d. 120
e. 250

Match the command with its response.

___ 8. Canned or fixed cycle for drilling — a. G01

___ 9. Canned or fixed cycle for the tapping cycle — b. G02

___ 10. Canned or fixed cycle for boring operations — c. G03

___ 11. End of program — d. G76

___ 12. Indicates counter-clockwise movement (circular interpolation) — e. G81

___ 13. Indicates clockwise movement (circular interpolation) — ab. G84

___ 14. Linear interpolation's preparatory function; commands the slides (lathe) or table (mill) to move the tool in a straight line from the current position to the command position — ac. M00 — ad. M05

___ 15. Program stop — ae. M30

Select the option which answers the question or completes the statement.

___ 16. What is a canned cycle?

a. Preset series of operations which direct machine tool movement to compete an action
b. Standardized routine which can be called up repeatedly
c. Group of commands
d. Single command that calls a group of commands
e. Preset point locations

___ 17. What is the purpose of using line numbers in a program?

a. The CNC controller always requires them.
b. Maintains an order for the operator to identify a certain line of information
c. Tells the programmer what line number he/she is on when writing the program manually
d. The tape-making device needs a line numbering system so it can keep track of each line of information internally.
e. None of the above

___ 18. What are preparatory functions?

a. Used only in metric work
b. Sometimes called "G" codes
c. Used only in inches
d. Do not apply in the absolute mode
e. Also called M codes

___ 19. Which of the following describes the absolute mode of dimensioning?

a. Best to use when accuracy and tolerance are unimportant
b. Determines the cutting tool location from a specific datum point (origin)
c. Most widely used of the two modes
d. Best mode for contouring work
e. Defines location in terms of finished product

___ 20. What kind of machine shop especially needs to keep a shop book?

a. One with many employees
b. A one-person operation
c. Any size
d. All of the above
e. None of the above

___ 21. Which is <u>not</u> true of a tool inventory sheet?

a. Corresponds with the tool cutter description on the manufacturing plan
b. Designates which position the tool is placed in the ATC
c. Compensates for undersized cutters
d. Deals with all aspects of tools except tool holders
e. All of the above

___ 22. Which describes the purpose of an operation set-up sheet?

a. Specifies how the workpiece is to be held to the table
b. Necessary when any new clamping or fixture is required
c. References the distance from home to part zero
d. All of the above
e. None of the above

___ 23. When might an operation set-up sheet be omitted?

a. In large, multi-operator shops
b. In shops where the operator has full responsibility for locating the workpiece on the work table
c. When the front view indicates the accumulated height of the fixture plus the workpiece thickness
d. When distances programmed to reference point returns and gage heights from the workpiece are described
e. Never

___ 24. What information is <u>not</u> commonly found on an operation instruction sheet?

a. Cutting tool diameter
b. Number of flutes
c. Distances programmed to reference point returns
d. Direction of spindle rotation
e. All of the above

___ 25. What is each tool change accompanied by?

a. Tool inventory sheet
b. Manufacturing plan
c. Operation set-up sheet
d. Operation instruction sheet
e. Title page

____ 26. List four skills or abilities a successful programmer must have.

a.

b.

c.

d.

NOTE: Completion of this test to specified score is the student's final evaluation for competency M4: Write program for CNC machine.

ANSWERS TO UNIT TEST (M4)

1. b
2. d
3. d
4. c
5. a
6. e
7. b
8. e
9. ab
10. d
11. ae
12. c
13. b
14. a
15. ac
16. a
17. b
18. b
19. b
20. a
21. d
22. d
23. b
24. c
25. d

26. Any four of the following:

    a. Strong knowledge of blueprint language and the ability to convert that language into an NC sketch
    b. Complete understanding of tolerances
    c. Ability to develop an adequate method for holding the workpiece during machining operations
    d. Ability to decide which machining operation should come first
    e. Ability to plan and work efficiently
    f. Ability to work safely
    g. Ability to think logically and methodically
    h. Ability to read and understand charts
    i. Basic geometry and trigonometry skills
    j. Capability to manipulate computer data
    k. Fluency in NC languages or formats
    l. Ability to prepare instructions
    m. Communication skills necessary to notify other departments or workers of significant change

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL
UNIT V: SETTING AND OPERATING THE NC MACHINE

UNIT OBJECTIVE

Upon completion of this unit, the student should understand NC set-up, machine preparation and operator's responsibilities. This knowledge will be evidenced by achieving the specified score on the assignment sheet and the unit test.

SPECIFIC OBJECTIVES

After completion of the lessons in this unit, the student should be able to:

1.1 Describe tools used in NC set-up.

1.2 Identify uses of the fixture in NC set-up.

1.3 Describe workpiece identification procedures. (competency M5)

2.1 Describe workpiece preparation procedures. (competency M2)

2.2 Explain tool length offset set-up.

2.3 Differentiate between true and false steps when loading program data.

3.1 Describe operator responsibilities while machining a workpiece. (competency M6)

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT V: SETTING AND OPERATING THE NC MACHINE

CONTENTS OF THIS UNIT

A. Objective sheet

B. References

C. Notes to the instructor

D. Lesson plans

1. Lesson 1: MACHINE SET-UP

   a. Information outline

   b. Handouts

      HO 1.1: Sample tooling form
      HO 1.2: Typical tool I.D. data sheet
      HO 1.3: Presetting the tool length

   c. Assignment sheet

      AS 1.1: Cutting tools

   d. Answers to assignment sheet

2. Lesson 2: OPERATING THE NC MACHINE

   a. Information outline

3. Lesson 3: MACHINING THE WORKPIECE

   a. Information outline

   b. Handout

      HO 3.1: Typical control panel

E. Unit test

F. Answers to unit test

REFERENCES

A. Books

1. Donaldson, Cyril, George H. LeCain and V. C. Goold. Tool Design. 3rd ed. New York: McGraw-Hill, 1973.

2. Luggen, William W. Fundamentals of Numerical Control. 2nd ed. Albany, NY: Delmar Publishers, Inc., 1988.

3. Roberts, Arthur D. Programming for Numerical Control Machines. 2nd ed. New York: McGraw Hill, 1978.

4. Seames, Warren S. Computer Numerical Control. 1st ed. Albany, NY: Delmar Publishers, Inc., 1986.

5. Stranton, George C. Numerical Control Programming. 1st ed. New York: John Wiley & Sons, Inc., 1988.

B. Others

1. Computerized Numerical Control (test item bank). Atlanta, GA: Vocational-Technical Education Consortium of States; Carbondale, IL: Dept. of Vocational Education Studies, Southern Illinois University, 1986.

2. Theories and Operation of Numerical Control/Computerized Numerical Control (NC/CNC) (U105001). Urbana: University of Illinois at Urbana-Champaign: Vocational Education Service, March 1988.

NOTES TO THE INSTRUCTOR

A. Suggested approaches to teaching this unit

1. Stress the need for understanding clamp positioning.

2. Read and interpret operator's manual. Although most NC machines are designed for low maintenance, encourage students to consult the operator's manual for care recommendations (competency M2). The amount of maintenance an operator/programmer does is also dependant on job duty restrictions.

3. Demonstrate the operation of setting "Z" tool length offset and loading parameter data.

B. Suggested student activities

1. Measure several reground tools and identify the correct amount of cutter compensation to the nearest standard-sized cutting tool.

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT V: SETTING AND OPERATING THE NC MACHINE

LESSON 1: MACHINE SET-UP

I. The tools **(HO 1.1-1.3)**

A. The NC machine requires considerable preparation before the workpiece can be machined. Various components, such as the cutting tools, fixture and raw work-stock, have to be brought together at the machine tool location.

B. The set-up person creates a tooling sheet based on data located in the tool inventory sheet of the shop book.

C. Proper tooling must be selected and placed in the sequence that it will be used.

D. Tooling must be transported safely to the machine tool area.

E. Cutting tool edges must be protected while in transit to specific CNC machines.

F. All cutting tools must be available at the machine tool before the program run begins. Missing tools should be reported to the programmer or specified personnel.

G. Cutting tools must be assembled with appropriate tool holders. The holder can be then inserted into the machine tool spindle or automatic tool changer (ATC).

H. Adapters and extensions are sometimes required and must be matched to the proper tool holder.

I. The set-up person checks the combined length of tool holder requirements on the tool inventory sheet.

J. The cutting tool can be tightened inside tool holder by various methods, depending upon the particular machine tool.

1. Tighten set screw in holder.

2. Tighten collet around holder with a wrench.

3. Tighten cutting tool in drill chuck using the chuck key.

K. Next, the tool must be inserted securely in the tool drum, turret or tool magazine in the order specified on the tool sheet.

L. Before inserting the tool into the automatic tool changer, the ATC must be wiped clean and free of chips, debris or foreign matter.

M. Different procedures are required for manual and automatic drawbar types of machines.

1. Manual type

a. Hold tool holder or adapter with one hand while operating drawbar.

b. Tighten drawbar to tool holder.

c. Place hand on spindle brake and insert wrench on drawbar head.

d. Tighten until firmly seated.

2. Automatic type

a. With spindle rotation off, grasp the tool holder and switch tool button to release position. Be prepared so that the tool holder will not slip from grip.

b. Place new tool holder or adapter assembly in spindle nose.

c. Align key slot between tool holder and spindle.

d. Engage drawbar by turning tool switch to hold position.

N. Some shops require the set-up person to set and adjust tool length compensation at this point.

II. The fixture

A. The NC machine is capable of machining a wide variety of workpieces. It would be impractical to create a new fixture for every workpiece.

B. The basic goal is to create the most universal fixture possible to reduce set-up time.

C. The fixture is designed to hold and produce several sized parts.

D. The set-up person has the job of physically transporting the fixture to the machine work station.

E. The fixture should be observed for damage or wear.

F. The machine tool table must be wiped clean of chips before the fixture is placed on it.

G. Identify and mount the correct fixture to the machine table.

H. After the work-holding device is placed on the table, it should be aligned to the machine axis and snugged down. Clamps should always be placed as close to the support as possible.

I. Place dial indicator in tool holder, collet or arbor in a position that puts the contact point in touch with the straight line, machined edge of the fixture.

J. Move table the length of the fixture parallel to the machine axis and observe indicator.

K. Watch dial indicator needle for movement and adjust the alignment of the work-holding device until the dial indicator remains constant.

L. Tighten work-holding device to table.

M. Indicate along the fixture's edge once again to verify the reliability of its alignment to the machine axis.

III. The workpiece

A. Often the workpiece has been machined on some tool room equipment prior to being sent to the CNC machine.

B. Properly identifying the workpiece at each stage of machining process is essential.

C. The workpiece is steel stamped or marked in some way to eliminate errors.

D. The set-up person must verify the workpiece's identity by cross-checking the blueprint identification number with the visible steel stamp or markings on the workpiece.

E. Place workpiece identification number on all paperwork, computer programs and workpiece storage bins where and if required.

F. Verification should also be made concerning the type of material and size of workpiece.

G. Compare type and size of material of each detail with the type of material shown on blueprint and/or operator's set-up sheet.

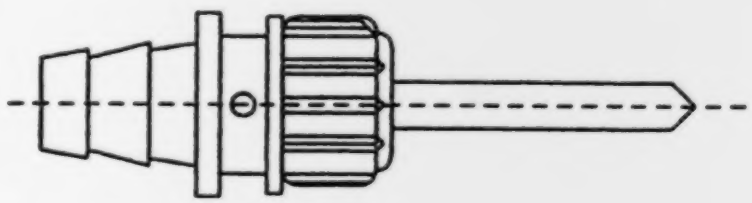
SAMPLE TOOLING FORM

PART NAME		PART NO.		DRAWING NO.	
MACHINE AND CONTROL		REVISION	PROG. BY	DATE	
SET UP AND TOOL INFORMATION			CHECK BY	PAGE 1 OF PAGES 1	
TO SET TLC TRAH POSITION					

Operation or Station Number	Tool			Remarks	Cut Speed	R.P.M.	Feed	CPT or FD/REV
	Description	Assembly No.	Set Dim.					

TYPICAL TOOL I.D. DATA SHEET

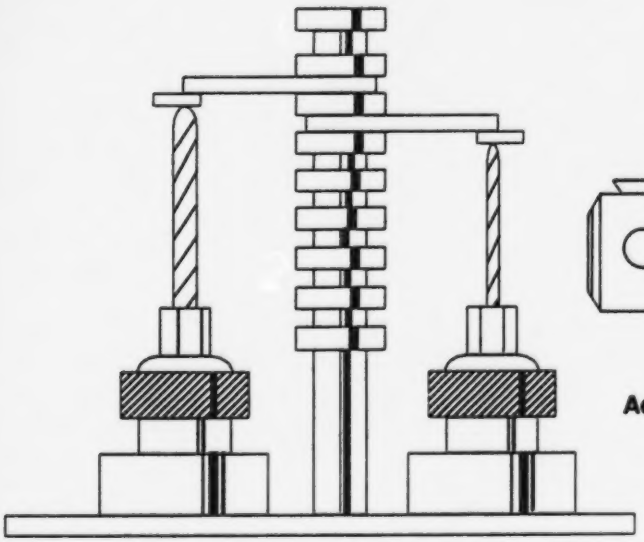
TOOL DATA SHEET	TOOL NAME
TOOL DRAWING	5/16" Drill (long)



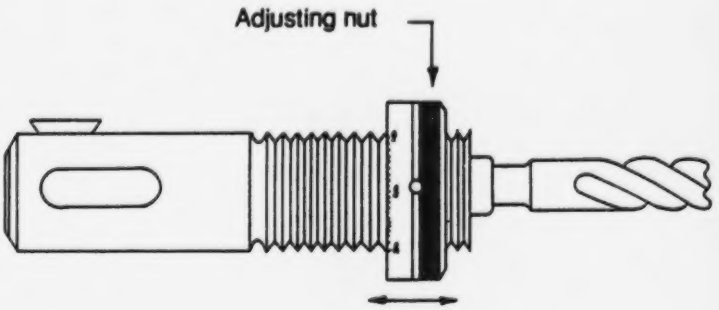
DURABLE TOOLING	TOOL IDENTIFICATION No.
91501 universal spring collect chuck	01121
901000-5/16 spring collect	DATE SET
901001 spring collect nut	1-1-86
	SET BY
	D.G.H.
PERISHABLE TOOLING	WHERE USED
5/16 extra length drill	327654
at least 7.5" flute lgth.	
9.0" O.A.L.	

TOOL NAME 5/16" Drill (long) TOOL I.D. NUMBER 01121

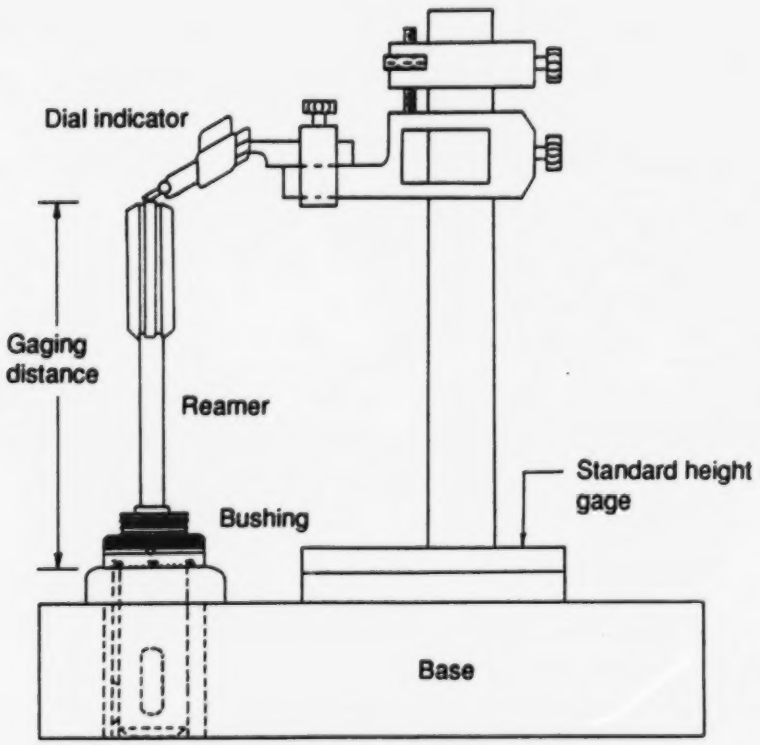
PRESETTING THE TOOL LENGTH



Universal presetting height gage



Adjustable spindle adapter (adjusted longitudinally)



Tool presetting with a standard height gage

CUTTING TOOLS

Place an "a" before true statements and a "b" before false ones.

____ 1. Machining can begin as long as tooling is available for the first segment of a multi-segment program.

____ 2. Cutting tool edges should be protected while in transit to CNC.

____ 3. Very little machine preparation is required when cutting tools are sharp.

____ 4. The set-up person must be able to read the shop book.

____ 5. A skilled programmer will assemble the cutters and tool holders.

____ 6. The ATC must be clean when inserting cutting tools.

____ 7. The set-up person's job is to position the preset length of tools.

____ 8. The cutter and adapter need not be handled when using an automatic type of drawbar tool changer.

____ 9. The set-up person never adjusts tool length compensation on the MCU.

____ 10. Some cutters are held by a collet, while other types are held by hand.

ANSWERS TO AS 1.1

1. b
2. a
3. b
4. a
5. b
6. a
7. a
8. b
9. b
10. b

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT V: SETTING AND OPERATING THE NC MACHINE

LESSON 2: OPERATING THE NC MACHINE

I. Preparing to machine workpiece

NOTE: Refer to the operator's manual for specific NC machine care and maintenance requirements. On the job, follow union or job duty guidelines for machine maintenance.

A. The operator first reviews the program in the shop book and verifies the manufacturing plan.

B. The tool inventory sheet is then matched with the physical tooling at the machine site.

C. In some shops it is the operator's responsibility to align the work-holding device with the machine axis.

D. Using the dial indicator, the operator should adjust the work-holding device to half the movement of the dial indicator needle.

E. Verify that workpiece material and program identification match.

F. Coolant reservoir must be replenished and machine lubrication levels restored to proper height.

G. Panel doors are to be closed and protective screens should be positioned around the machine if necessary.

H. Turn on air compressor and attach air lines (if required).

I. Locate the master control switch on control or machine panel and position control switch to ON.

J. Inspect table, quill, cross-slide and tool drum position for possible collision.

K. If no obstructions are found, turn on the MCU or jog the machine tool in all directions to assure that all slides move freely.

L. Check the operation of the coolant pump by activating the coolant controls.

M. Observe control panel for start position message display. A typical message would be X-Y coordinates.

II. Setting tool length offset

A. Tool length offset is the distance from the part or gage height to the bottom of the tool when located at Z home position.

B. This Z-axis distance must be registered for each tool in its allocated parameter setting.

C. A simple, quick method of doing this is to insert the cutting tool and tool holder in the machine spindle and zero return the spindle to its fully retracted position.

D. In the handle or jog mode, move the bottom of the cutter close to the workpiece surface with Z-axis movement.

E. Slide a .010" shim back and forth between the workpiece surface and the bottom of the cutter.

F. Using caution, very slowly continue lowering the cutting tool until the shim drags while being pulled. Do not lower too far because this can damage the workpiece, cutting tool or machine spindle.

G. The distance displayed on the control monitor screen is an accurate tool length offset when .010" (the thickness of the shim) is added.

H. Load this dimension in the tool length offset area on the parameter section of the menu. See manufacturer's recommendations for directions on how parameter setting is done.

I. Repeat this process for each cutting tool used in the program.

III. Loading program data from storage

A. Key-in program on machine.

1. Refer to part print specifications.

2. Assign program number.

3. Using the manually written manuscript, enter all information. (See manufacturer's operating manual for specific machine tool.)

B. Call up program in Direct Numerical Control.

1. The first step, after turning on the computer, is to enter the code name or number to access information banks.

2. Call up the directory of programs available. This lists the scheduled programs to be run on this machine.

3. Enter program name or number needed.

4. Check program in computer.

a. Check for condition of program.

b. Contact CNC programmer if program is not available.

5. Download data from microcomputer to MCU.

C. Traditionally, binary tape has been a common method of storing program data.

1. Set tape load switch to OFF.

2. Place tape on reels, on spools or between reading head and cover.

3. Check alignment of tape and adjust the roller guides.

4. Engage tape with spindle drive pins and clamps.

5. Press tape load switch to take up tape slack.

MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT V: SETTING AND OPERATING THE NC MACHINE

LESSON 3: MACHINING THE WORKPIECE

I. Operator responsibility **(HO 3.1)**

A. Dry-run program to check program for errors.

1. Activate the machine and load program data from storage.

2. Set cutter depths so that cutter will not hit workpiece during dry run.

3. Watch cutter travel during dry run to check program. Be prepared to stop machine if program malfunction occurs or cutter appears to be on a collision course with workpiece, clamps or fixture.

4. If no problems occur, return to "home" and rewind tape to begin running first segment of program.

B. While watching the first segment machine, the operator may need to adjust the speed and/or feed rate.

1. Determine the amount of feed and spindle rpm increase or decrease.

2. Most machine control units have variable control override switches that increase speed and feed rate by percentages. If no percentage switch is available, the operator may have to key-in a new spindle speed. (See manufacturer's programming manual for specific machine tool options available.)

3. On the operation instruction sheet, the operator must attach a note or highlight any area of the cutter path where it was necessary to alter the programmed speed or feed rate.

4. Observe chip and cutter response to altered speed and/or feed rate.

C. Stop spindle at appropriate time, visually inspecting both workpiece and cutting tool.

D. The cutting tool must be removed and replaced if worn or dull.

E. When removing holder and cutting tool as one unit, use the following procedure.

1. Remove tool holder from machine.

2. Remove cutting tool from tool holder.

3. Clean and insert new cutting tool.

4. Install tool holder with cutting tool in machine. (See manufacturer's specification.)

F. When removing cutting tool from holder (using indexable inserts of a qualified tool), use the following steps.

1. Leave holder in machine.

2. Disengage cutting tool from holder (see manufacturer's tooling manual).

3. Clean tool holder before inserting.

4. Insert new cutter.

G. If the substitute cutting tool used to replace the original cutter has been reground (resharpened and therefore slightly undersized), the operator must enter this information into the program.

H. Measure the replacement tool and allow for the difference between actual and programmed cutter diameter. When compensating for a different sized cutter, -1.0000, +1.0000 is an allowable difference.

I. Enter the cutter compensation code valve on the MCU (see specifics in the operator's manual).

J. The operator must run the entire program or block-by-block execution of program without interruption or intervention.

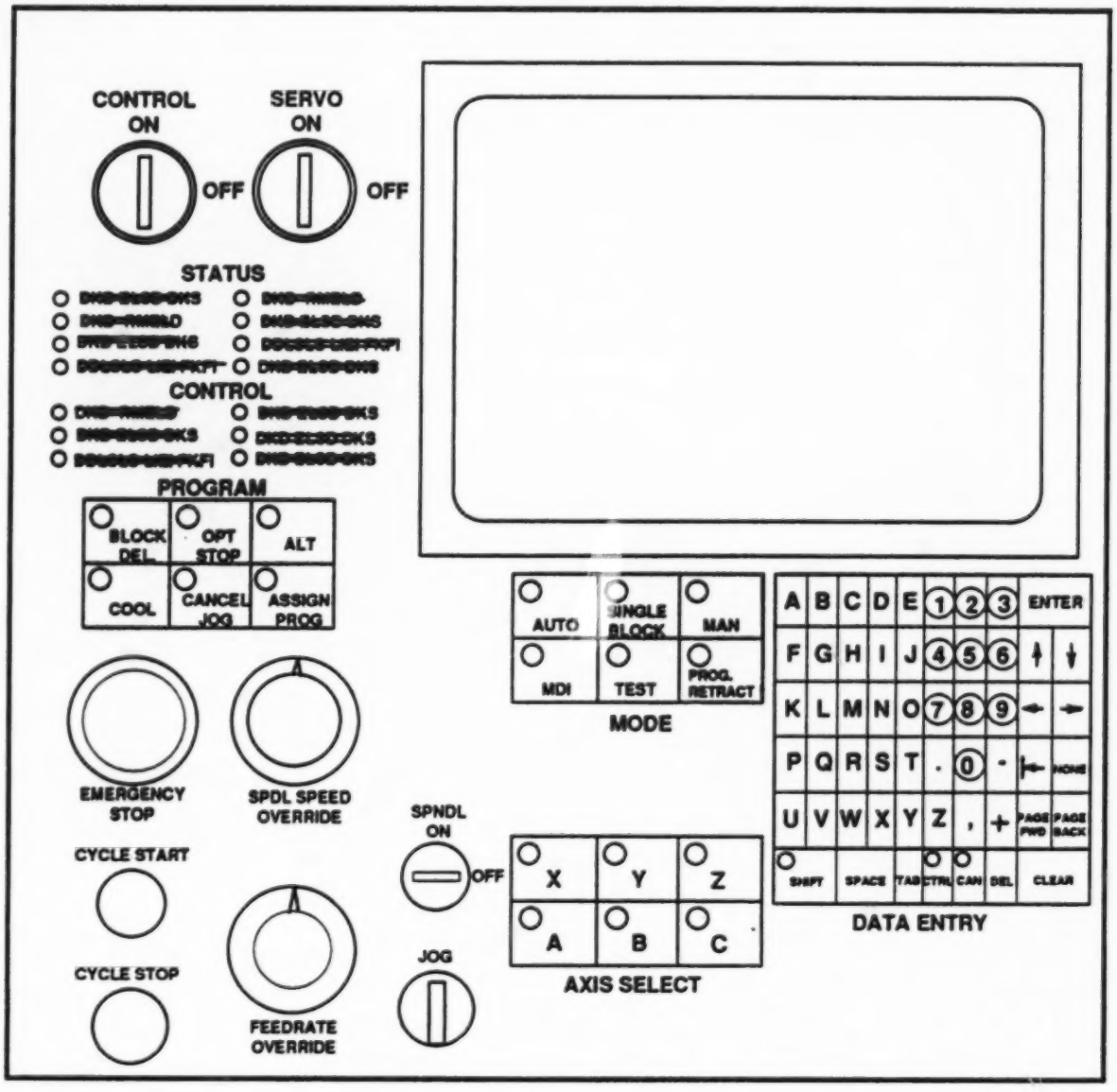
K. When necessary to stop the machining operation, the operator should know clearly whether an immediate or gradual stop is appropriate.

1. Emergency stop and reset buttons can be used for an immediate machine spindle and table travel halt.

2. The signal block switch permits completion of the present cutting operation and waits for the signal to continue.

CAUTION: See machine tool builder's operator manual for stop and start procedures--adhere to them with great alertness. Information stored in the bubble memory may be dumped when the emergency stop button is pressed. If this is the case, the MCU may "think" the cutter is in a different position than it truly is.

TYPICAL CONTROL PANEL



MODULE: (CNC) COMPUTERIZED NUMERICAL CONTROL

UNIT V: SETTING AND OPERATING THE NC MACHINE

UNIT TEST

Select the option which answers the question or completes the statement.

___ 1. The basic function of a typical fixture is to locate and secure what tool or part?

a. Tool holder
b. Workpiece
c. Tool
d. Spindle
e. Dial indicator

___ 2. When located at Z home position, what is the distance from the part or gage height to the bottom of the tool?

a. Tool length dimension
b. Tool length path
c. Tool length offset
d. Tool length compensation
e. Tool size

___ 3. What should clamps be placed close to?

a. Tool
b. Support
c. X axis
d. Set point
e. Path

___ 4. What should be done if one tool is missing when setting up the tools for multiple tool usage?

a. Omit the step that involves that tool.
b. Substitute a similar tool.
c. Cancel the operation and return the program to the programmer.
d. Report the missing tool information to the programmer or specified personnel.
e. None of the above

___ 5. Which describes procedure for transporting tools from storage to the work area?

a. A tool carrier is needed for five or more tools.
b. A tool carrier is not required for roughing tools.
c. Each tool should be examined and placed in the tool carrier to protect the cutting edge.
d. A tool carrier is required for finish-cut tools only.
e. None of the above

___ 6. Which of the following is <u>not</u> a sound tooling practice?

a. Check all tools before they are used.
b. Clean tools before using.
c. Always choose the longest tool length.
d. Dispose of damaged or defective tools.
e. Transport tools carefully.

___ 7. While loading a tool drum/turret, you notice a dull tool. What is the next step?

a. Reduce feed rate.
b. Increase the spindle speed.
c. Increase spindle speed and reduce the feed rate if the tool is not chipped.
d. Replace the tool after the job in progress is finished.
e. Return the tool for replacement.

___ 8. What tool is used to manually tighten the drawbar head?

a. Operator's hand
b. Drawbar wrench
c. Drawbar jog mode
d. Drawbar automatic tightening button
e. Adjustable wrench

___ 9. What is the first step in operating an automatic drawbar?

a. Engage selector switch.
b. Align keyways in tool-holder or adapter with keyways in the spindle nose.
c. Place a new tool-holder in the spindle nose.
d. Clean holder and pocket of chips, dirt and other foreign materials.
e. None of the above

___ 10. When compensating for a different sized cutter, what is the allowable difference between the programmed diameter and the actual diameter?

a. -0.5000, +0.5000
b. -1.0000, +1.0000
c. -1.5000, +1.5000
d. -1.7500, +1.7500
e. -2.0000, +2.0000

___ 11. What is a workpiece mounting fixture designed to reduce?

a. Part set-up time
b. Machining operations
c. Finishing operations
d. Transfer time
e. Machining time

___ 12. When preparing to mount the workpiece in the fixture, what should the operator do first?

a. Test-run the machine.
b. Clamp a scrap workpiece in fixture.
c. Clean chips from work table and fixture.
d. Mount fixture to back of work table.
e. None of the above

___ 13. What does the workpiece detail identification number cross-reference to?

a. Program
b. Programmer
c. Blueprint
d. Operator
e. Parts list

___ 14. What is the result if the dial indicator needle does not move as it is traversed the length of the work-holding device?

a. Indicator is broken
b. Work-holding device is 1" out of alignment
c. Work-holding device is in alignment
d. Quill has been retracted
e. Workpiece is properly mounted

___ 15. After checking the alignment of a work-holding device, the dial indicator needle moves. How should the device be adjusted?

a. Half the movement of the dial indicator needle
b. To the full movement of dial indicator needle
c. It should <u>not</u> be adjusted.
d. In increments of .050"
e. To one-fourth the needle's movement

___ 16. What aspects of a workpiece should be verified before each stage of machining?

a. Type of material
b. Size of material
c. ID number
d. All of the above
e. None of the above

___ 17. To set tool length offset, what sized shim is used between the workpiece surface and bottom of the cutter?

a. .100"
b. .200"
c. .010"
d. .020"
e. .030"

___ 18. When loading program data from any storage media, which of the following cannot be a step?

a. Adjust tool offset length.
b. Download data from microcomputer to MCU.
c. Load binary tape.
d. Call up program in DNC.
e. Ensure MCU and storage media compatibility.

___ 19. What is done first when calling up a program in DNC?

a. Call up directory of programs.
b. Enter password to access data bank.
c. Select program number or name.
d. Check condition of program.
e. Check number of operators using the program.

20. List three steps in preparing to machine a workpiece.

a.

b.

c.

NOTE: Completion of this test to specified score is the student's final evaluation for competencies M5 and M6.

ANSWERS TO UNIT TEST

1.	b	(M5)	11.	a	(M5)
2.	c	(M5)	12.	c	(M5)
3.	b	(M5)	13.	c	(M5)
4.	d	(M5)	14.	c	(M5)
5.	c	(M5)	15.	a	(M5)
6.	c	(M5)	16.	d	(M5)
7.	e	(M6)	17.	c	(M5)
8.	b	(M5)	18.	a	(M6)
9.	d	(M5)	19.	b	(M6)
10.	b	(M6)			

20. Any three of the following: (M2)

a. Consult operator's manual for specific NC machine care and maintenance.
b. Review the program in the shop book and verify the manufacturing plan.
c. Match the tool inventory sheet with the physical tooling at the machine site.
d. In some shops it is the operator's responsibility to align the work-holding device with the machine axis.
e. Using the dial indicator, the operator should adjust the work-holding device to half the movement of the dial indicator needle.
f. Verify that workpiece material and program identification match.
g. Coolant reservoir must be replenished and machine lubrication levels restored to proper height.
h. Panel doors are to be closed and protective screens should be positioned around the machine if necessary.
i. Turn on air compressor and attach air lines (if required).
j. Locate the master control switch on control or machine panel and position control switch to ON.
k. Inspect table, quill, cross-slide and tool drum position for possible collision.
l. If no obstructions are found, turn on the MCU or jog the machine tool in all directions to assure that all slides move freely.
m. Check the operation of the coolant pump by activating the coolant controls.
n. Observe control panel for start position message display.